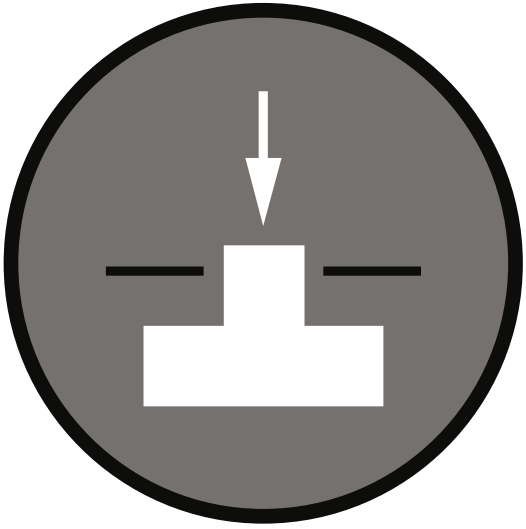


SVSOLID™

2D / 3D Stress Deformation Modeling Software

Tutorial Manual

Written by:
Murray Fredlund, PhD, PEng
Gilson Gitirana, PhD
Robert Thode, BSc,GE

Edited by:
Murray Fredlund, PhD, PEng

SoilVision Systems Ltd.
Saskatoon, Saskatchewan, Canada

Software License

The software described in this manual is furnished under a license agreement. The software may be used or copied only in accordance with the terms of the agreement.

Software Support

Support for the software is furnished under the terms of a support agreement.

Copyright

Information contained within this Tutorial Manual is copyrighted and all rights are reserved by SoilVision Systems Ltd. The SVSOLID software is a proprietary product and trade secret of SoilVision Systems. The Tutorial Manual may be reproduced or copied in whole or in part by the software licensee for use with running the software. The Tutorial Manual may not be reproduced or copied in any form or by any means for the purpose of selling the copies.

Disclaimer of Warranty

SoilVision Systems Ltd. reserves the right to make periodic modifications of this product without obligation to notify any person of such revision. SoilVision does not guarantee, warrant, or make any representation regarding the use of, or the results of, the programs in terms of correctness, accuracy, reliability, currentness, or otherwise; the user is expected to make the final evaluation in the context of his (her) own problems.

Trademarks

Windows™ is a registered trademark of Microsoft Corporation.
SoilVision® is a registered trademark of SoilVision Systems Ltd.
SVOFFICE ™ is a trademark of SoilVision Systems Ltd.
SVFLUX ™ is a trademark of SoilVision Systems Ltd.
CHEMFLUX ™ is a trademark of SoilVision Systems Ltd.
SVHEAT ™ is a trademark of SoilVision Systems Ltd.
SVAIRFLOW ™ is a trademark of SoilVision Systems Ltd.
SVSOLID ™ is a trademark of SoilVision Systems Ltd.
SVSLOPE ™ is a trademark of SoilVision Systems Ltd.
ACUMESH ™ is a trademark of SoilVision Systems Ltd.
FlexPDE® is a registered trademark of PDE Solutions Inc.

Copyright © 2008
by
SoilVision Systems Ltd.
Saskatoon, Saskatchewan, Canada
ALL RIGHTS RESERVED
Printed in Canada

1 Introduction 4
2 A Two-Dimensional Example Model 5
2.1 Model Setup 6
2.2 Results and Discussion 15
3 A Three-Dimensional Example Model 17
3.1 Model Setup 18
3.2 Results & Discussion 27
4 2D: Edge Drop of a Flexible Impervious Cover 29
4.1 Model Overview 29
4.2 Initial SVFLUX Seepage Analysis 30
4.2.1 Model Setup 30
4.2.2 Results & Discussion 35
4.3 SVFLUX Transient Seepage Analysis 36
4.3.1 Model Setup 36
4.3.2 Results & Discussion 42
4.4 SVSOLID Stress/Deformation Analysis 44
4.4.1 Model Setup 44
4.4.2 Results & Discussion 50
5 References 55

1 Introduction

The Tutorial Manual serves a special role in guiding the first time users of the SVSOLID software through a typical example problem. The example is "typical" in the sense that it is not too rigorous on one hand and not too simple on the other hand.

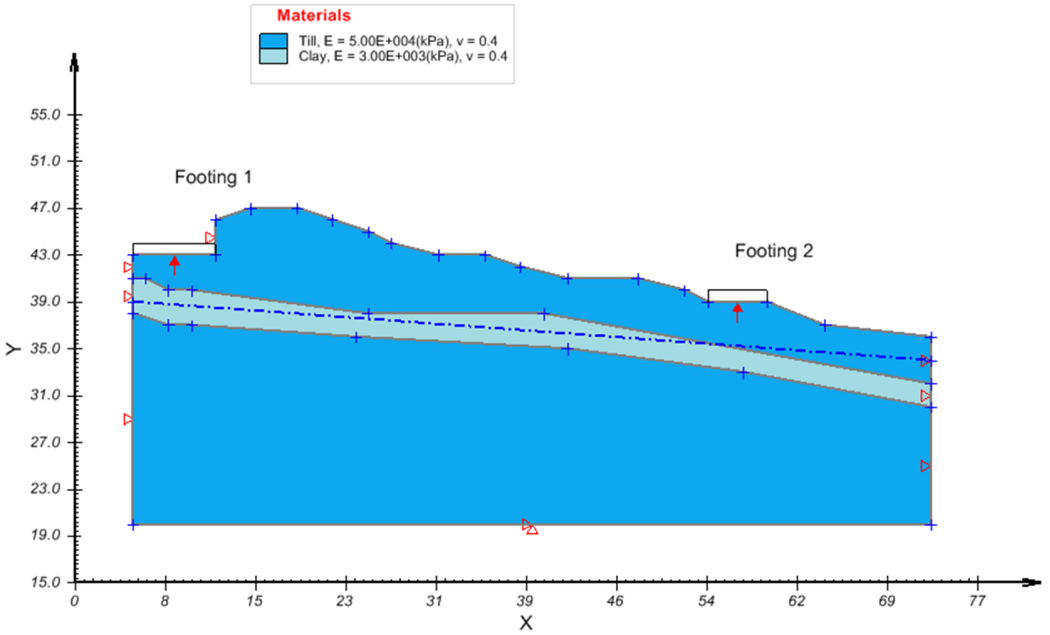
The Tutorial Manual serves as a guide by: i) assisting the user with the input of data necessary to solve the boundary value problem, ii) explaining the relevance of the solution from an engineering standpoint, and iii) assisting with the visualization of the computer output. An attempt has been made to ascertain and respond to questions most likely to be asked by first time users of SVSOLID.

2 A Two-Dimensional Example Model

The following example introduces some of the features included in SVSOLID. The example problem sets up a model of a simple slope with two foundation loads applied. A water table is present. The purpose of this model is to determine the stress conditions in the slope due to applied loads and the magnitude of displacement under each footing. The model dimensions and material properties are provided below.

Project: Foundations
Model: Tutorial2D
Minimum authorization required: STUDENT

Model Description and Geometry



Ground			Seam			Water Table	
X	**Y**		**X**	**Y**		**X**	**Y**
5	41		5	38		5	39
5	38		8	37		73	34
5	20		10	37			
73	20		24	36			
73	30		42	35			
73	32		57	33			
73	36		73	30			
64	37		73	32			
59	39		40	38			
54	39		25	38			
52	40		10	40			

48	41		8	40			
42	41		6	41			
38	42		5	41			
35	43						
31	43						
27	44						
25	45						
22	46						
19	47						
15	47						
12	46						
12	43						
5	43						

Material Properties

Material 1: Till

Young's Modulus, E = 50000 kPa
Poisson's Ratio, ν = 0.4
Initial Void Ratio, e_o = 1
Vertical Body Load, γ_y = -21 kN/m^3

Material 2: Clay

Young's Modulus, E = 3000 kPa
Poisson's Ratio, ν = 0.4
Initial Void Ratio, e_o = 1
Vertical Body Load, γ_y = -18.5 kN/m^3

2.1 Model Setup

In order to set up the model described in the preceding section, the following steps will be required. The steps fall under the general categories of:

a. Create model
b. Enter geometry
c. Specify boundary conditions
d. Apply material properties
e. Specify model output
f. Run model
g. Visualize results

a. Create Model

Since FULL authorization is required for this tutorial, the user must perform the following steps to ensure full authorization is activated:

1. Plug in the USB security key,
2. Go to the *File > Authorization* dialog on the SVOFFICE Manager,
3. Software should display full authorization. If not, it means that the security codes provided by SoilVision Systems at the time of purchase have not yet been entered. Please see the the Authorization section of the SVOFFICE User's Manual for instructions on entering these codes.

The following steps are required to create the model:

1. Open the *SVOFFICE Manager* dialog,
2. Create a new project called Tutorial by pressing the *New* button next to the list of projects,
3. Create a new model called UserTutorial2D by pressing the *New* button next to the list of models. The new model will be automatically added under the recently created Tutorial project,
4. Select the following:

   Application: SVSOLID

   System: 2D Vertical

   Type: Steady-State

   Units: Metric

The user should also set the World Coordinate System to ensure that the model will fit in the drawing space. The World Coordinate System settings can be set under the *World Coordinate System* tab on the *Create Model* dialog.

1. Access the World Coordinate System tab on the *Create New Model* dialog,
2. Enter the World Coordinates System coordinates shown below into the dialog,

   x-minimum: 0

   y-minimum: 15

   x-maximum: 77

   y-maximum: 57

3. Click *OK* to close the dialog.

The workspace grid spacing needs to be set to aid in defining region shapes. The geometry of the model has coordinates of a precision of 1m. In order to effectively draw the geometry with this precision when using the mouse, the grid spacing must be set to a maximum value of 1.

1. The *View Options* dialog should open once the *Create Model* dialog is closed,
2. Enter 1 for both the horizontal and vertical grid spacing,
3. Click *OK* to close the dialog.

Options must be selected here in order to specify a water table as the initial pore-water pressure conditions. Drawing of the water table is explained later in this tutorial.

1. Open the *Settings* dialog by selecting *Model* > *Settings* in the workspace menu,
2. Select "Consider PWP" as the Analysis option,
3. Press *OK* to close the dialog,
4. Open the *Initial Conditions* dialog by selecting *Model* > *Initial Conditions* > *Settings*,
5. Move to the *Pore-Water Pressure* tab,
6. Select "Draw Water Table" as the Initial PWP Option,
7. Click *OK* to close the dialog.

b. Enter Geometry

A region in SVSOLID forms the basic building block for a model. A region represents both a physical portion of material being modeled and a visualization area in the SVSOLID CAD workspace. A region forms a geometric shape that define the material boundaries. Also, other modeling objects including features, water tables, text, and line art can be defined on any given region.

The model being used in this tutorial is divided into three regions, which are named Ground, Seam, and Water Table. The first two regions will have one of the materials previously defined specified as its material properties. The third region will be used to represent the material below the water table. To add the necessary regions follow these steps:

1. Open the *Regions* dialog by selecting *Model* > *Geometry* > *Regions* from the menu,
2. Change the first region name from Region 1 to Ground. To do this, highlight the name and type the new text,
3. Press the *New* button to add a second region,
4. Change the name of the second region to Seam,
5. Select "Clay Shale" as the material for the Seam region,
6. Click *New* to add the third region,
7. Name the third region Water Table,
8. Click *OK* to close the dialog.

The shapes that define each material region will now be created. Note that when drawing a geometric shape, information will be added to the region that is current in the Region Selector. The Region Selector is at the top of the workspace.

- **Define the Ground**

The instructions below explain the use of the mouse to create the ground region.

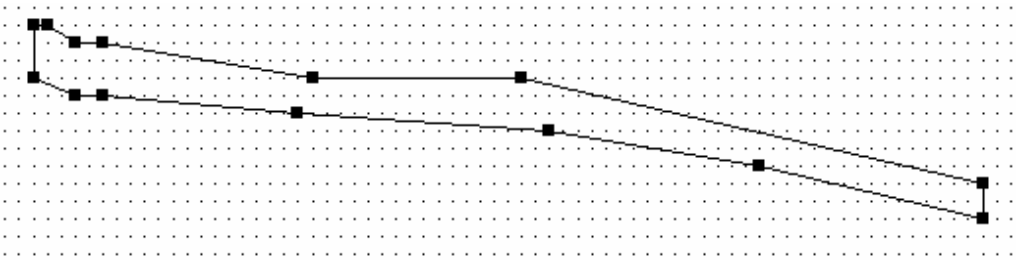
1. Select "Ground" as the region by going to *Model* > *Geometry* > *Regions*, and clicking on "Ground",
2. Press *OK* to close the dialog,
3. Select *Draw* > *Model Geometry* > *Polygon Region* from the menu,
4. The cursor will now be changed to cross hairs,
5. Move the cursor near to coordinates (5,41) in the drawing space. You can view the coordinates for the current position of the mouse in the status bar at the bottom right of the screen,
6. To select a point as part of the desired region shape, left click on the point,
7. Now move the cursor near (5,38) and then left click on the point. A line is now drawn from coordinates (5,41) to (5,38),
8. Refer to the geometry table at the beginning of this tutorial and add the remaining points,
9. To add the last point, move the cursor near the coordinate point (5,43) and right click snapping the cursor to the point. Then double-click on the point to finish the shape. A line is now drawn from (12,43) to (5,43) and the shape is automatically completed in SVSOLID by drawing a line from (5,43) back to the starting point, (5,41),

- **Define the Seam**

The instructions below explain the use of the mouse to create the seam region.

10. Select "Seam" as the region by going to *Model* > *Geometry* > *Regions*, and clicking on "Seam",
11. Press *OK* to close the dialog,
12. Select *Draw* > *Model Geometry* > *Polygon Region* from the menu,
13. The cursor will now be changed to cross hairs,
14. Move the cursor near to coordinates (5,38) in the drawing space. You can view the coordinates for the current position of the mouse in the status bar just below the workspace,
15. To select the point as part of the desired region shape, left click on the point,
16. Now move the cursor near to coordinates (8,37) and then left click on the point. A line is now drawn from coordinates (5,38) to (8,37),
17. Refer to the geometry table at the beginning of this tutorial and add the remaining points,
18. To add the last, point, move the cursor near the point (5,41) and right click snapping the cursor the coordinate point. Double-click on the point to finish the shape. A line is now drawn from (6,41) to (5,41) and the shape is automatically completed in SVSOLID by drawing a line from (5,41) back to the starting point, (5,38).

If the seam geometry has been entered correctly the shape should look as follows:



- **Define the Water Table**

The instructions below explain the use of the mouse to create the water table region.

1. Select "Water Table" as the region by going to *Model* > *Geometry* > *Regions*, and clicking on "Water Table",
2. Press *OK* to close the dialog,
3. Select *Draw* > *Model Geometry* > *Water Table* from the menu,
4. The cursor will now be changed to cross hairs,
5. Move the cursor near to coordinates (5,39) in the drawing space. You can view the coordinates for the current position of the mouse in the status bar at the bottom right of the screen,
6. To select a point as part of the desired region shape, left click on the point,
7. Now move the cursor near (73,34) and then double click on the point. A line is now drawn from coordinates (5,39) to (73,34).

After all the region geometries have been entered, the diagram will appear as shown at the beginning of this tutorial.

c. Specify Boundary Conditions

Boundary conditions must be applied to all region points. The starting point for that particular boundary condition is initiated at any boundary point on a region geometry. The boundary condition will then extend over subsequent line segments around the edge of the region. The direction for the application of the boundary conditions is determined by the way the geometry was originally entered. Boundary conditions remain in effect around a geometry shape until they are re-defined. The user may not define two different boundary conditions over the same line segment.

More information on boundary conditions can be found in *Menu System* > *Model Menu* > *Boundary Conditions* > *2D Boundary Conditions* of the User's Manual.

The next step is to specify the boundary conditions. A load expression needs to be defined each of the footing locations on the ground region. The sides should be fixed in the *X*-direction. At the base the region should fixed in both the *X* and *Y* directions. The Seam region is internal to the Ground region and will not need to be altered

as far as boundary conditions are concerned. The steps in specifying the boundary conditions are as follows:

1. Select the "Ground" region in the drawing space,
2. Select *Model* > *Boundaries* > *Boundary Conditions* from the menu. The *Boundary Conditions* dialog will open,
3. Select the coordinate point (5,41) from the list on the *Segment Boundary Conditions* tab,
4. From the *X* Boundary Condition drop-down select a "Fixed Boundary Condition",
5. From the *Y* Boundary Condition drop-down select a "Free Boundary Condition",
6. Enter the remaining Boundary Conditions found in the Boundary Condition Summary table below,
7. Click the *OK* button to close the dialog.

NOTE:

The Fixed *X* Boundary Condition for the coordinate point (5,41) becomes the boundary condition for the following line segments that have a Continue Boundary Condition until a new boundary condition is specified. By specifying a Free condition at point (73,36) the Continue Boundary Condition is turned off.

- **Boundary Condition Summary**

X	Y	X Boundary Condition	Y Boundary Condition
5	41	Fixed	Free
5	38	Continue	Continue
5	20	Continue	Fixed
73	20	Continue	Free
73	30	Continue	Continue
73	32	Continue	Continue
73	36	Free	Continue
64	37	Continue	Continue
59	39	Continue	Load Expression = -100
54	39	Continue	Free
52	40	Continue	Continue
48	41	Continue	Continue
42	41	Continue	Continue
38	42	Continue	Continue
35	43	Continue	Continue
31	43	Continue	Continue
27	44	Continue	Continue
25	45	Continue	Continue
22	46	Continue	Continue
19	47	Continue	Continue
15	47	Continue	Continue
12	46	Fixed	Continue

12	43	Free	Load Expression = -80
5	43	Fixed	Free

d. Apply Material Properties

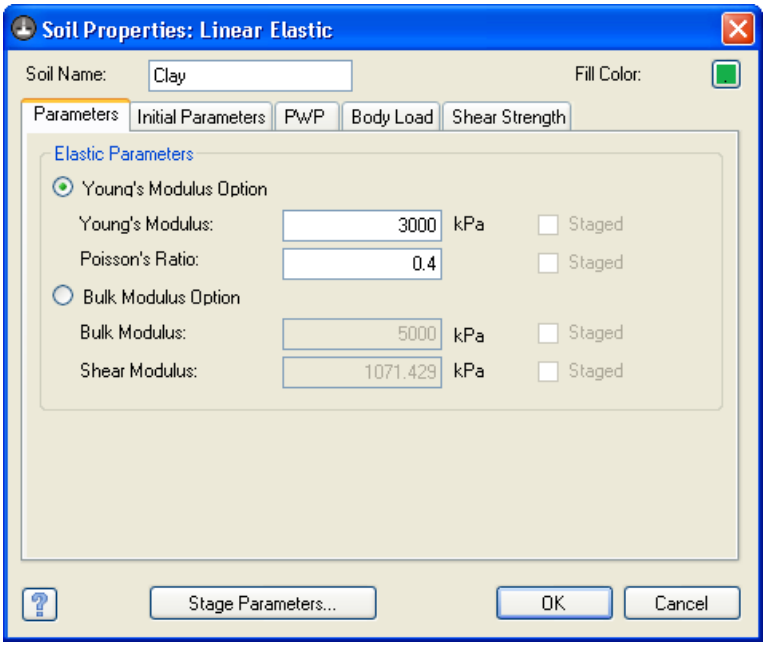
The next step in defining the model is to enter the material properties for the two materials that will be used in the model. This section provides instructions on creating the "clay shale" material. Repeat the process to add the other material.

1. Open the *Materials* dialog by selecting *Model* > *Materials* > *Manager* from the menu,
2. Click the *New* button to create a material. A unique Material Index is generated that is used to reference the material in other *SVSOLID* dialogs,
3. Enter "Clay Shale" for the material name in the dialog that appears,

NOTE:

When a new material is created, you can specify the display color of the material by using the Fill Color box in the Material Properties menu. This material color will be displayed for any region that has a material assigned to it will display that material's fill color.

4. Select the new material and click "Properties" to open the *Material Properties* dialog,
5. Move to the *Parameters* tab,
6. Enter the Young's Modulus value of 3000 kPa,



7. Enter the Poisson's Ratio value of 0.4,
8. Move to the *Initial Parameter* tab,
9. Enter a Void Ratio value of 1,
10. Move to the *Body Load* tab,
11. Enter the *X*-Axis Body Load as 0 kN/m3,
12. Enter the *Y*-Axis Body Load as –18.5 kN/m3,
13. Press *OK* to close the dialog,
14. Repeat these steps to create the "till" material; refer to the data provided under the "A Two Dimensional Example Model" section at the beginning of this tutorial,
15. Press *OK* to close the *Materials Manager* dialog.

NOTE:
The negative sign for the body load indicates that the vertical body load acts in a downward direction.

Once all material properties have been entered, we must apply the materials to the corresponding regions.

1. Open the *Region Properties* dialog by selecting *Model* > *Geometry* > *Region Properties* from the menu,
2. Select "Clay Shale" as the material for the Seam region,
3. Select "Till" as the material for the Ground region,
4. Press the *OK* button to accept the changes and close the dialog.

e. Specify Model Output

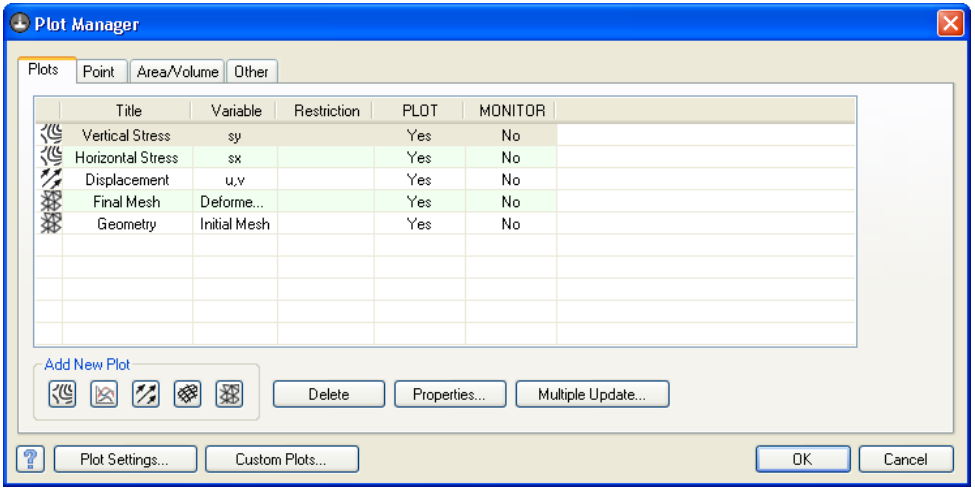
Two levels of output may be specified: i) output (graphs, contour plots, fluxes, etc.) which are displayed during model solution, and ii) output which is written to a standard finite element file for viewing with ACUMESH software. Output is specified in the following two dialogs in the software:

i) Plot Manager:	Output displayed during model solution.
ii) Output Manager:	Standard finite element files written out for visualization in ACUMESH or for inputting to other finite element packages.

PLOT MANAGER

There are numerous graphical plots that can be specified to visualize the results of the model. A few typical graphs will be generated for this tutorial example model. These plots are the solution finite element mesh, horizontal and vertical, stress contours, and displacement vectors.

1. Open the *Plot Manager* dialog by selecting *Model* > *Reporting* > *Plot Manager* from the menu,



2. The toolbar at the bottom left corner of the *Plot Manager* dialog contains a button for each plot type. Clicking to the *Contour* button will begin adding the first contour plot. The *Properties* dialog drop-down will open,
3. Enter the title Vertical Stress,
4. Select s_y as the variable to plot from the drop-down,
5. Click the *Properties* button and go to the *Output Options* tab in the *Plot Properties* dialog. Select the "PLOT" output option,
6. Click *OK* to close the dialog and add the plot to the list of requested graphical plots,
7. Repeat Steps 2 to 6 to create the plots as shown above,
8. Click the *Plot Settings* button to open the *Plot Settings* dialog,
9. Open the *Resolution* tab and enter a Magnification Factor of 50. This will allow the computed deformation to be plotted to an exaggerated scale, making them easier to examine,
10. Click *OK* to close the *Plot Manager* and return to the workspace.

OUTPUT MANAGER

1. Open the *Output Manager* dialog by selecting *Model* > *Reporting* > *Output Manager* from the menu,
2. The toolbar at the bottom left corner of the dialog contains a button for each output file type. Click on the *ACUMESH* button to begin adding the output file, The *Output File Properties* dialog will open,
3. Click *OK* to close the dialog and add the output file to the list,
4. Click *OK* to close the *Output Manager* and return to the workspace.

f. Run Model

The current model may be run by selecting the *Solve* > *Run* menu option.

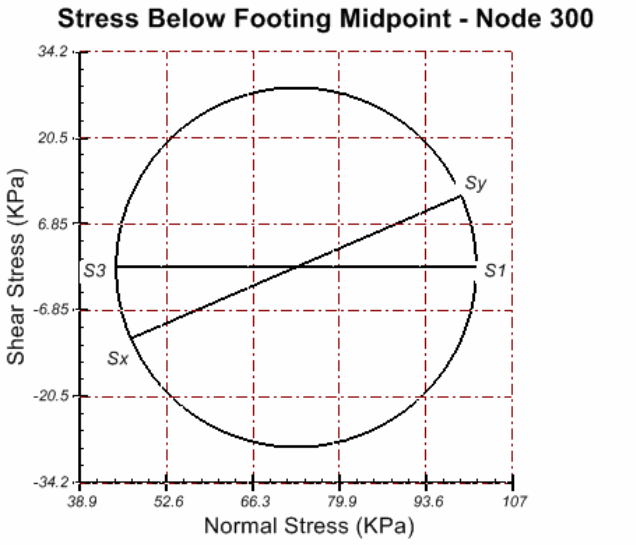
g. Visualize Results

The visual results for the current model may be examined by selecting the *View* > *ACUMESH* menu option.

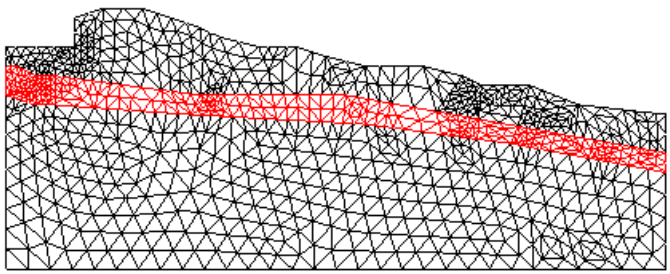
2.2 Results and Discussion

After the computations for the model have been completed, the results will be displayed as a series of thumbnail plots within the SVSOLID solver. Right-clicking the mouse on any thumbnail plot and selecting "Maximize" will enlarge that particular graph. The following sections will give a brief description of each plot that can be generated.

The ACUMESH 2D visualization software can be used for improved graphics quality and a greater range of plotting options. Mohr Circle type plots of the principle stresses can be generated for any selected node in the finite element mesh as shown below.

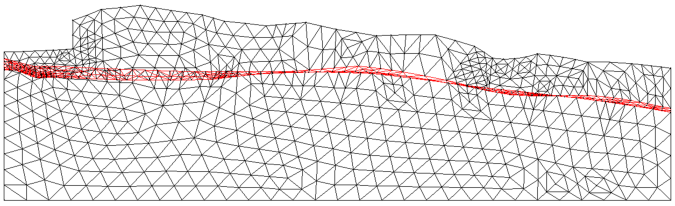


- **Solution Mesh**



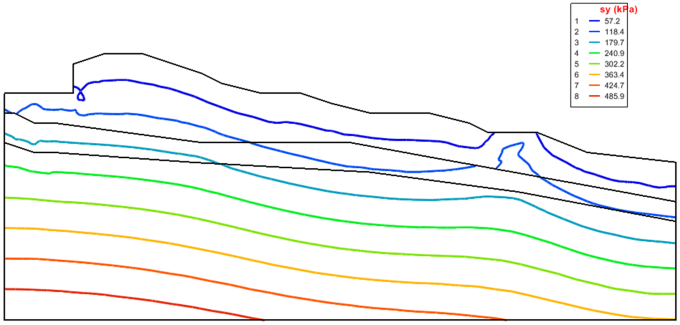
The Mesh plot displays the finite-element mesh generated by the solver. The mesh is automatically refined in critical areas such as directly beneath the footings where there is a greater influence from the applied loads.

- **Deformed Mesh**



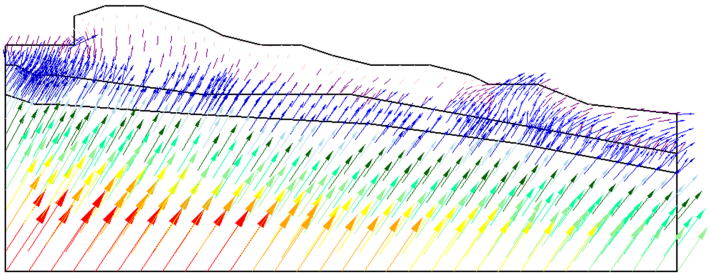
The displacements in this plot have been magnified by 50 times. Note that the greatest displacement occurs directly beneath the footings. Also the displacements in the clay seam are much greater than the surrounding till due to the differences in Young's Modulus.

- **Vertical Displacements**



A stress bulb is generated beneath each footing due to each footing load. The body load of both materials generates the overall stress state.

- **Displacement Vectors**



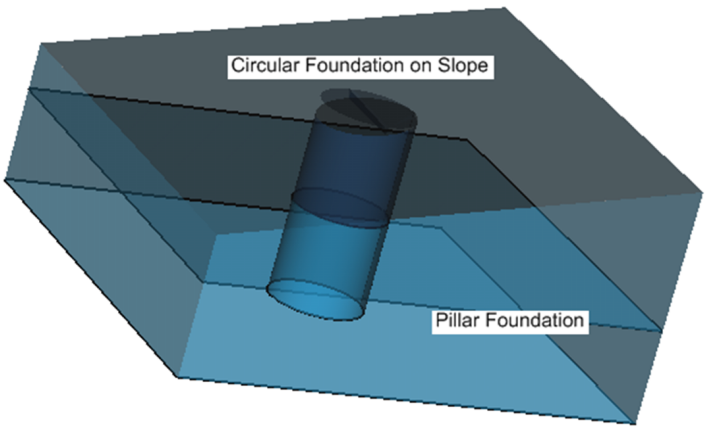
Displacement Vectors show both the direction and the magnitude of the displacement at specific points in the model. The lower Young's Modulus in the clay shale seam result in greater displacements than in the overlying till. The maximum displacement of 0.16m (16mm) occurs beneath Footing 1 where the load is greater and the distance to the clay seam is less.

3 A Three-Dimensional Example Model

The following example introduces you to three-dimensional modeling using SVSOLID. The model computes the stress and displacement generated as a result of placing pillar foundation on a sloping ground surface. The circular pillar foundation has been placed at the mid-slope. The model is modeled using two regions, three surfaces, and two materials. The model data and material properties are provided below. This model is set up to run with the Student Version of SVSOLID.

Project: Foundations
Model: Tutorial3D
Minimum authorization required: STUDENT

Model Description and Geometry



Slope				Pillar	
X	**Y**			**X**	**Y**
0	0		Center:	10	10
20	0				
20	20		Radius:	2	
0	20				

Material Properties

Material 1: Till

Data Type: Linear Elastic
Young's Modulus, E = 10,000 kPa
Poisson's Ratio, $\nu = 0.4$
Initial Void Ratio, $e_o = 1$
Vertical Body Load, $\gamma_y = -21$ kN/m^3

Material 2: Concrete

Data Type: Linear Elastic
Young's Modulus, E = 29,580,000 kPa
Poisson's Ratio, $\nu = 0.2$
Initial Void Ratio, $e_o = 0$
Vertical Body Load, $\gamma_y = -23.5$ kN/m^3

3.1 Model Setup

In order to set up the model described in the preceding section, the following steps will be required. The steps fall under the general categories of:

a. Create model
b. Enter geometry
c. Specify boundary conditions
d. Apply material properties
e. Specify model output
f. Run model
g. Visualize results

a. Create Model

The following steps are required to create the model:

1. Open the *SVOFFICE Manager* dialog,
2. Select "ALL" under the Applications combo box and "ALL" for the Model Origin combo box,
3. Select the project called UserTutorial from the list of projects,
4. Create a new model called UserTutorial3D by pressing the New button next to the list of models. The new model will be automatically added under the recently created UserTutorial project. Use following settings when creating a new model:

   Application: SVSOLID
   System: 3D Vertical
   Type: Steady-State
   Units: Metric

The user should also set the World Coordinate System to ensure that the model will fit in the drawing space. The World Coordinate System settings can be set under the *World Coordinate System* tab on the *Create Model* dialog.

1. Access the *World Coordinate System* tab on the *Create Model* dialog,
2. Enter the World Coordinates System coordinates shown below into the dialog,

   x-minimum: 0

y - minimum: -5

z - minimum: -5

x - maximum: 25

y - maximum: 25

z - maximum: 10

3. Click *OK* to close the dialog.

The workspace grid spacing needs to be set to aid in defining region shapes. The geometry data for this model has coordinates of a precision of 1m. The grid spacing should therefore be set to a maximum of 1 in order to effectively draw the geometry with this precision using the mouse

1. The *View Options* dialog should open once the *Create Model* dialog is closed,
2. Enter 1 for both the horizontal and vertical spacing,
3. Click *OK* to close the dialog.

b. Enter Geometry

A region in SVSOLID is the basic building block for a model. A region represents both the material being modeled and a visualization area in the SVSOLID CAD workspace. A region will have geometric shapes that define its material boundaries. Also, other modeling objects including features, water tables, text, and line art can be defined on any given region.

This model is divided into two regions, which are called the Slope and the Pillar. Each region has one materials specified as its material properties. The regions and materials can be combined using the following steps:

1. Open the *Regions* dialog by selecting *Model* > *Geometry* > *Regions* from the menu,
2. Change the first *region name* from R1 to Slope. This can be done by highlighting the name and typing new text,
3. Press the *New* button to add a second region,
4. Change the name of the second region to Pillar,
5. Click *OK* to close the dialog.

- **Define the Slope region**

1. Select "Slope" as the region by going to *Model* > *Geometry* > *Regions* and clicking on "Slope",
2. Select *Draw* > *Model Geometry* > *Region Polygon* from the menu,
3. The cursor will now be changed to cross hairs,
4. Move the cursor near (0,0) in the drawing space. The coordinates of the current position of the mouse can be viewed on the status bar just below the workspace,
5. To select the point as part of the shape left click on the point,

6. Now move the cursor near (20,0). Right click to snap the cursor to the exact point and then left click on the point. A line is now drawn from (0,0) to (20,0),
7. Now move the cursor near (20,20). Right click to snap the cursor to the exact point and then left click on the point,
8. For the last point (0,20), right click to snap the cursor to the point. Double-click on the point to finish the shape. A line is now drawn from (20,20) to (0,20) and the shape is automatically finished by SVSOLID by drawing a line from (0,20) back to the starting point, (0,0),

NOTE:
If a mistake is made in creating the geometry, then select a shape with the mouse and select *Edit* > *Delete* from the menu. This will remove the entire shape from the region. To edit the shape use the *Region Properties* dialog.

- **Define the Pillar**

9. Ensure that "Pillar" is current in the region selector,
10. Select *Draw* > *Model Geometry* > *Region Circle* from the menu,
11. The cursor will now be changed to cross hairs,
12. Move the cursor near (10,10) in the drawing space. The coordinates of the current position the mouse can be seen in the status bar just below the drawing space.
13. To select a point as part of the shape, hold the left click button on the *point*.
14. Drag the cursor in a *X* or *Y* direction and release the left click button once the mouse is 2m away from the initial point. This will create a 2 meter radius for the circle.

NOTE:
At times it may be difficult to snap to a grid point that is near a line defined for a region. In this case, turn, the object snap off by clicking on *"OSNAP"* in the status bar.

This model consists of three surfaces with differing dimensions and grid densities. By default, every model initially has two surfaces.

- **Define Surface 1**

This surface is already present so the next step is to define the grid lines.

1. Select "Surface 1" by going to *Model* > *Geometry* > *Surfaces* and clicking on "Surface 1",
2. Press *OK* to close the dialog,
3. Select *Model* > *Geometry* > *Surface Properties* in the menu to open the *Surface Properties* dialog,
4. Select "Elevation Data" from the Definition Options drop-down,

5. Select the *Elevations* tab and click the *Define Grid* button to set up the grid for the selected surface,
6. There will be default grid lines of 0 and 10 present. Click the *Add Regular* button to open the Add Regular X gridlines dialog,
7. Enter -5 for *Start*, 5 for Increment Value, and 25 for End,
8. Click "Add" to add the gridlines and close the dialog,
9. Move to the *Y Grid Lines* tab and repeat steps 4 to 6 for the *Y* gridlines,

Elevations must be specified for all the grid points. Now that the "grid" has been set up:

10. Select "Surface 1" in the Surface Selector,
11. Select *Model* > *Geometry* > *Surface Properties* in the menu to open the *Surface Properties* dialog,
12. Select the *Elevations* tab,
13. Enter 0 in the Set Nulls field,
14. Click on the *Set Nulls* button and all the missing elevations will be set to 0.

- **Define Surface 2**

This surface is already present. The extent of this grid are smaller than for Surface 1 and the grid is denser. The Surface 2 grid also has different densities in the X and Y directions.

15. Select "Surface 2" in the Surface Selector,
16. Select *Model* > *Geometry* > *Surface Properties* in the menu to open the *Surface Properties* dialog,
17. Select "Elevation Data" from the Definition Options drop-down,
18. Select the *Elevations* tab and click the *Define Grid* button to set up the grid for the selected surface,
19. There will be default grid lines of 0 and 10 present. Click the *Add Regular* button to open the *Add Regular X Gridlines* dialog,
20. Enter 0 for Start, 2 for Increment Value, and 20 for End,
21. Click "Add" to add the gridlines and close the dialog,
22. Move to the *Y Grid Lines* tab,
23. There will be default grid lines of 0 and 10. Click the *Add Regular* button to open the *Add Regular Y Gridlines* dialog,
24. Enter 0 for Start, 4 for Increment Value, and 20 for End,
25. Click "Add" to add the gridlines and close the dialog,

Now that the grid has been set up, elevations must be specified for all the grid points:

26. Select "Surface 2" in the Surface Selector,
27. Select *Model* > *Geometry* > *Surface Properties* in the menu to open the *Surface Properties* dialog,
28. Enter 4 in the Set Nulls field,
29. Click the *Set Nulls* button and all the missing elevations will be set to 4m,

- **Define Surface 3**

Follow these steps to add the third surface to the model.

30. To open the *Surface* dialog you may select *Model* > *Geometry* > *Surfaces* from the menu,
31. Click the *New* button to open the *Insert Surfaces* dialog,
32. Enter 1 as the Number of New Surfaces,
33. Select to place the new surface At The Top,
34. Select "Copy Grid" From An Existing Surface,
35. Select "Surface 2" from the drop-down,
36. Choose to Exclude the elevations,
37. Press *OK* to add the surface,

Surface 3 and its grid has now been added. The next step is to provide the elevation values. The geometry will be generated using the 3D Plane Interpolation method:

38. Select "Surface 3" in the Surface Selector,
39. Select *Model* > *Geometry* > *Surface Properties* in the menu to open the *Surface Properties* dialog,
40. Select the "Elevation data" from the Surface Definitions Options drop-down and select the *Elevations* tab to set up the elevations for Surface 3.
41. Select point (0,0),
42. Enter a *Z* elevation of 6,
43. Select point (0,20),
44. Enter a *Z* elevation of 6,
45. Select point (20,20),
46. Enter a *Z* elevation of 10,
47. Press the *3D Plane Interpolation* button and press the *Yes* button,
48. Press *OK* to close the dialog.

c. Specify Boundary Conditions

Boundary conditions must be applied at all points around a region. Once a boundary condition is applied to a boundary point this defines the starting point for that particular boundary condition. The boundary condition will then extend over subsequent line segments around the edge of the region in the direction in which the region shape was originally entered. Boundary conditions remain in effect around a shape until re-defined. The user may not define two different boundary conditions over the same line segment.

More information on boundary conditions can be found in *Menu System* > *Model Menu* > *Boundary Conditions* > *2D Boundary Conditions* in the User's Manual.

Now that all of the regions, surfaces, and materials have been successfully defined, the next step is to specify the boundary conditions on the region shapes. The vertical boundaries of the slope will be fixed as will the base. A load of 500 kPa can be applied to the top of the pillar. The steps for specifying the boundary conditions are as follows:

- **Slope Region**
    1. Make sure your model is being viewed in 2D. This option is available to the left side of the workspace by clicking the *2D* button,
    2. Select the "Slope" region by going to *Model* > *Geometry* > *Regions* and clicking on "Slope",
    3. Select "Surface 1" by going to *Model* > *Geometry* > *Surfaces* and clicking on "Surface 1",
    4. From the menu select *Model* > *Boundaries* > *Boundary Conditions*. The *Boundary Conditions* dialog will open and display the boundary conditions for Surface 1. These boundary conditions will extend from Surface 1 to Surface 2 over Layer 1,
    5. Select the *Surface Boundary Conditions* tab at the top of the dialog,
    6. From the *X* Boundary Condition drop-down select a "Fixed" Boundary Condition,
    7. From the *Y* Boundary Condition drop-down select a "Fixed" Boundary Condition,
    8. From the *Z* Boundary Condition drop-down select a "Fixed" Boundary Condition,
    9. Select the *Segment Boundary Conditions* tab at the top of the dialog,
    10. From the *X* Boundary Condition drop-down for the first point, select a "Fixed" Boundary Condition,
    11. From the *Y* Boundary Condition drop-down for the first point, select a "Fixed" Boundary Condition,

NOTE:
The *Fixed* boundary condition for the point (0,0) becomes the boundary condition for the following line segments that have a *Continue* boundary condition applied until a new boundary condition is specified.

The boundary conditions for the slope region are to be the same for Layer 2 as for Layer 1. Therefore, the Surface 1 segment boundary conditions can be copied to Surface 2:

12. In the *Boundary Conditions* dialog ensure that Surface 1 is currently in the drop-down,
13. Press the *Copy Boundary Conditions* button to open the *Copy Boundary Conditions* dialog,
14. Select "Surface 2" from the list,
15. Press *OK*,
16. Close the *Boundary Conditions* dialog,

- **Pillar Region**

17. Select the “Pillar” region in the region selector,
18. Select "Surface 3" in the surface selector,
19. From the menu select *Model* > *Boundaries* > *Boundary Conditions*. The *Boundary Conditions* dialog will open and display the boundary conditions for Surface 3.
20. Select the *Surface Boundary Conditions* tab at the top of the dialog,
21. From the *Z* Boundary Condition drop-down select a Load Expression boundary condition,
22. Enter a value of –500 in the expression field,
23. Click *OK* to close the dialog.

d. Apply Material Properties

The next step in defining the model is to enter the material properties for the two materials comprising the model. The slope consists of a till material and the pillar foundation is concrete. This section will provide instructions on inputting data for the till material. Repeat the process to add the other material.

1. Open the *Materials Manager* dialog by selecting *Model* > *Materials* > *Manager…* from the menu,
2. Click the "*New...*" button to create a material,

NOTE:
When a "new" material is created, you can specify the display color of the material using the Fill Color box on the Material Properties menu. The color for the soil will be displayed for any region that has a material assigned.

3. Enter 'Till' for the material name and Linear Elastic as data type, then press *OK*,
4. The *Material Properties* dialog will automatically open,
5. Move to the *Parameters* tab,
6. Enter a Young’s Modulus value of 10000 kPa,
7. Enter a Poisson’s Ratio value of 0.4,
8. Move to the Initial Parameters tab,
9. Enter the Initial Void Ratio value of 1,

10. Move to the *Body Load* tab,
11. Enter the *X*-Axis Body Load as 0 kN/m3,
12. Enter the *Y*-Axis Body Load as 0 kN/m3,
13. Enter the *Z*-Axis Body Load as –21 kN/m3,
14. Press *OK* to close the dialog,
15. Repeat the above steps to input the properties for concrete. Refer to the data provided under the "A Three-Dimensional Example Model" section at the beginning of this tutorial,
16. Press *OK* to close the *Materials Manager* dialog.

NOTE:
The negative sign for the body load indicates that the vertical body load will act in a downward direction.

Each region will cut through all the layers in a model creating a separate "block" in each layer. Each block can be assigned a material or left as void. A void area is assumed to be an air space. In this model, all "blocks" will be assigned a material.

1. Select "Slope" in the Region Selector,
2. Select *Model* > *Materials* > *Material Layers* from the menu to open the *Material Layers* dialog,
3. Select the "Till" material from the drop-down for Layer 2,
4. Select the "Till" material from the drop-down for Layer 1,
5. Close the dialog using the *OK* button,
6. Select "Pillar" in the Region Selector,
7. Select *Model* > *Materials* > *Material Layers* from the menu to open the *Material Layers* dialog,
8. Select the "Concrete" material from the drop-down for Layer 2,
9. Select the "Till" material from the drop-down for Layer 1,
10. Close the dialog using the *OK* button.

e. Specify Model Output

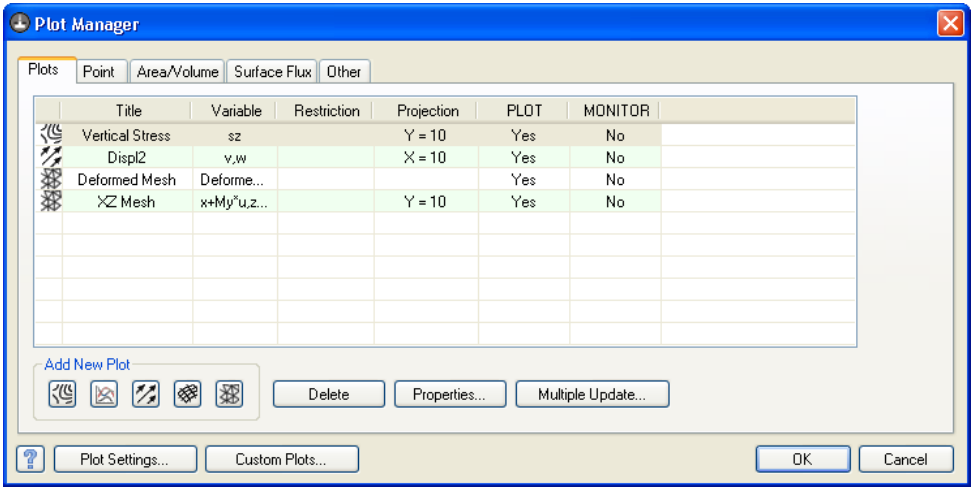
Two levels of output may be specified: i) output (graphs, contour plots, fluxes, etc.) which are displayed during model solution, and ii) output which is written to a standard finite element file for viewing with ACUMESH software. Output is specified in the following two dialogs in the software:

i) Plot Manager: Output displayed during model solution.
ii) Output Manager: Standard finite element files written out for visualization in ACUMESH or for inputting to other finite element packages.

PLOT MANAGER

There are numerous types of plots that can be specified to visualize the computed results from the model. A few plots will be generated for this tutorial example model including a plot showing vertical stress contours, the deformed mesh, and displacement vectors.

1. Open the *Plot Manager* dialog by selecting *Model* > *Reporting* > *Plot Manager* from the menu,



2. The toolbar at the bottom left corner of the dialog contains a button for each plot type. Click on the *Contour* button to begin adding the first contour plot. The *Plot Properties* dialog will open,
3. Enter the title Vertical Stress,
4. Select s_z as the variable to plot from the drop-down for the contour plot of the vertical stress,
5. Move to the *Projection* tab,
6. Select "Plane" as the Projection Option,
7. Select *Y* from the Coordinate Direction drop-down,
8. Enter 10 in the Coordinate field. This will generate a 2D slice at Y = 10m on which the stress contours will be plotted,
9. Move to the *Output Options* tab. Select the "PLOT output" option,
10. Click *OK* to close the dialog and add the plot to the list,
11. Repeat steps 2 to 10 to create the plots shown in the following screen-shot above,
12. Click *OK* to close the Plot Manager and return to the workspace.

OUTPUT MANAGER

1. Open the *Output Manager* dialog by selecting *Model* > *Reporting* > *Output Manager* from the menu,
2. The toolbar at the bottom left corner of the dialog contains a button for each output file type. Click on the *ACUMESH* button to begin adding the output file. The *Output File Properties*

dialog will open,

3. Enter the title ACUMESHOut,
4. Click *OK* to close the dialog and add the output file to the list,
5. Click *OK* to close the *Output Manager* and return to the workspace.

f. Run Model

The model is now ready for the analysis to be performed. Select *Solve* > *Analyze* from the menu. This action will write a descriptor file and open the SVSOLID solver. The solver will automatically begin solving the model.

g. Visualize Results

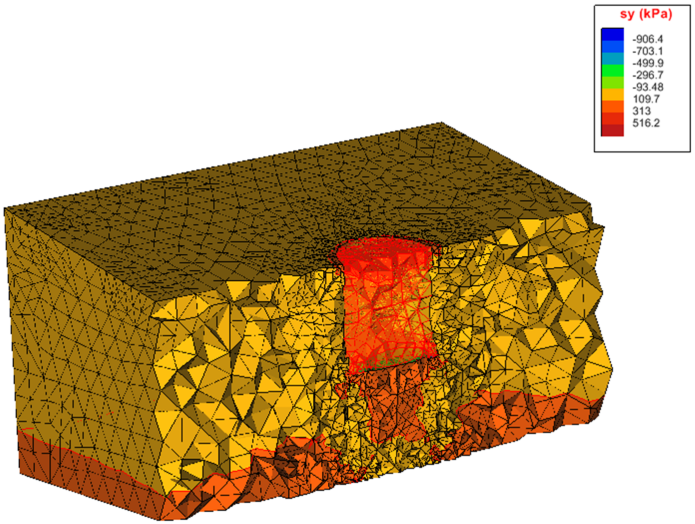
The results for the current model may be visualized by selecting the Open ACUMESH: *View* > *ACUMESH* menu option.

3.2 Results & Discussion

After the computations are complete, the results will be displayed using the dialog of thumbnail plots within the SVSOLID solver. It is possible to right-click the mouse and select the "Maximize" to enlarge any of the thumbnail plots. This section will give a brief description of each plot that was generated.

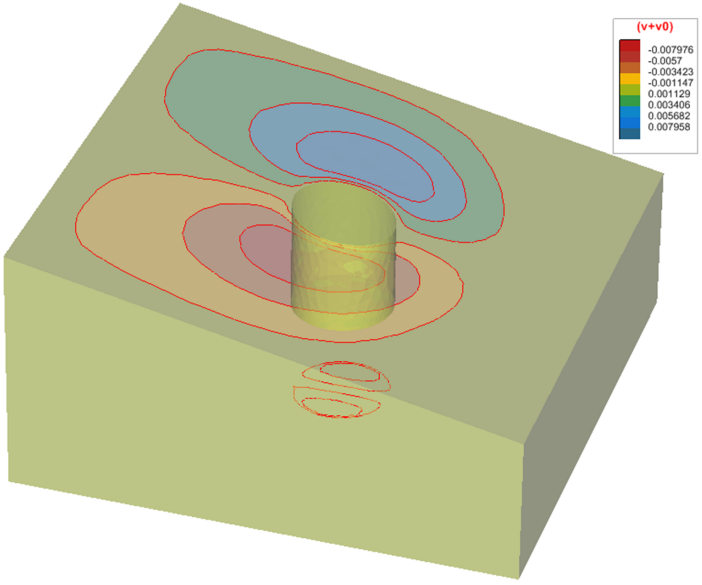
When the computations associated with the analysis are complete, it is possible to also visualize output plots using ACUMESH. In order to view plots in ACUMESH, select *Window* > *ACUMESH* from the menu.

- **Stress Contours**



The stress state generated by the load on the pillar can be examined using the ACUMESH software. The effects of both vertical load and skin friction can be observed.

- **Displacement Contours**



The above figure shows the displacement vectors in the direction of movement. The magnitude of the displacement at specific points in the model is also shown. The pillar displaces 0.07m (7.0mm) due to the 500 kPa load.

4 2D: Edge Drop of a Flexible Impervious Cover

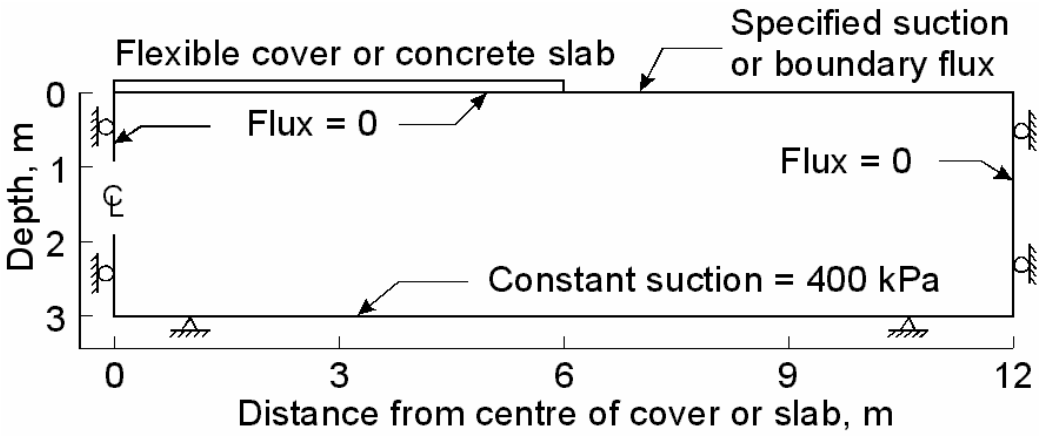
This example simulates the modeling of water infiltration into a material and the subsequent ground movements that are likely to occur. A 2D Edge Drop simulation is performed for a Flexible Impervious Cover.

4.1 Model Overview

The following example will provide a step by step guide to modeling slab movement using a manual iteration technique involving the SVFLUX and SVSOLID software packages. There are two main scenarios that are commonly considered; namely the Edge Drop of a slab caused by shrinking of the material due to evaporation (i.e., increase in soil suction) and Edge Lift of a slab caused by swelling due to infiltration (i.e., decrease in soil suction). This tutorial model will consider the Edge Drop scenario.

Model Description and Geometry

A 12m wide flexible impervious cover in 2D is considered. Since the cover is symmetrical the portion to the right of the centerline will be modeled. A material region 3m deep and 12m wide is used.



Material Properties

Stress Material Properties	Values
Coefficient of permeability at saturation, k_{sat}	1x10-8 m/s
Volumetric water content at saturation, θ_s	0.45
Parameters for SWCC (Fredlund & Xing, 1994) and	a = 300 kPa
permeability function (Leong and Rahardjo, 1997)	n = 1.5
	m = 1
	h_r = 3000 kPa
	Sat. Suction = 0.1 kPa
	p = 1

Stress Material Properties	Values
Total Unit Weight, gt	17.2 kN/m2
Initial Void Ratio, eo	1
Swelling Index, Cs	0.15
Swelling Index, Cm	0.13
Poisson's Ratio, ms	0.4
Coefficient of earth pressure at rest, Ko	0.33

Solution Outline

The manual iteration method involves a number of steps to arrive at the final displacements. These example models are included in the SVFLUX and SVSOLID model files for reference under the Project Name of SlabOnGround. The Model Name is indicated in parentheses.

1. Initial SVFLUX seepage analysis (Shrink_Initial),
2. SVFLUX transient seepage analysis (Shrink_Transient),
3. SVSOLID stress/deformation analysis (Shrink_Day5).

4.2 Initial SVFLUX Seepage Analysis

The purpose of the initial SVFLUX seepage analysis is to get an initial head profile to use as initial conditions for the SVFLUX transient seepage and to get an initial pore-water pressure profile to use as initial conditions for the SVSOLID stress/deformation analyses.

Project: SlabOnGround
Model: Shrink_Initial

4.2.1 Model Setup

In order to set up the model described in the preceding section, the following steps will be required. The steps fall under the general categories of:

a. Create model
b. Enter geometry
c. Specify boundary conditions
d. Apply material properties
e. Specify model output
f. Run model
g. Visualize results

a. Create Model

The following steps are required to create the model:

1. Open the *SVOFFICE Manager* dialog,
2. Select "ALL" under the Applications combo box and "ALL" for the Model Origin combo box,
3. Select the project called UserTutorial from the list of projects,
4. Create a new model called UserED_Initial by pressing the *New* button next to the list of models. The new model will be automatically added under the recently created UserTutorial project,
5. Select the following:

   Application: SVFLUX

   System: 2D Vertical

   Type: Steady-State

   Units: Metric

The user should also set the World Coordinate System to ensure that the model will fit in the drawing space. The World Coordinate System settings can be set under the *World Coordinate System* tab on the *Create Model* dialog.

1. Access the *World Coordinate System* tab on the *Create New Model* dialog,
2. Enter the World Coordinates System coordinates shown below into the dialog,

   x-minimum: -5

   y-minimum: -8

   x-maximum: 17

   y-maximum: 5

3. Click *OK* to close the dialog.

b. Enter Geometry

The shape that defines the material region will now be created. Note that when drawing geometric shapes the region that is current in the region selector is the region the geometry will be added. The Region Selector is at the top of the workspace.

Region Geometry

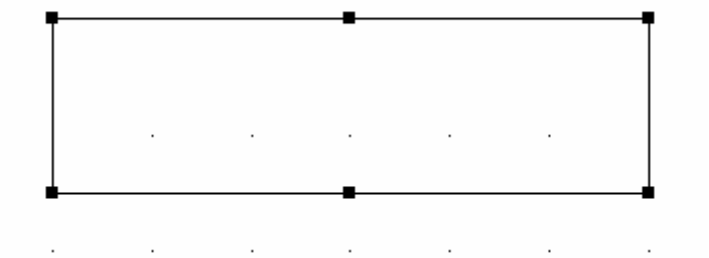
X	Y
0	-3
6	-3
12	-3
12	0
6	0
0	0

The ground shape can be drawn using the mouse or the data points can be pasted into the *Region Properties*

dialog.

1. Open the *Regions* dialog by selecting *Model* > *Geometry* > *Regions* from the menu,
2. Change the first region name from R1 to Ground. This can be done by highlighting the name and typing new text,
3. Select *Draw* > *Model Geometry* > *Polygon Region* from the menu,
4. The cursor will now be changed to cross hairs. Move the cursor near (0,-3) in the drawing space. You can view the coordinates for the current position of the mouse in the status bar just below the drawing space,
5. To select the point as part of the shape left click on the point,
6. Now move the cursor near (6,-3) and then left click on the point. A line is now drawn from (0,-3) to (6,-3),
7. Refer to the geometry table above and add the remaining points,
8. To add the last point, move the cursor near the point (0,0) and right-click snapping the cursor to the point. Double-click on the point to finish the shape. A line is now drawn from (6,0) to (0,0) and the shape is automatically completed by SVSOLID with a line from (0,0) back to the start point, (0,-3).

If the geometry has been entered correctly the shape should look as follows:



NOTE:

If a mistake was made during the input of the coordinate points for a shape, select a shape with the mouse and select *Edit* > *Delete* from the menu. This will remove the entire shape from the region. To edit the shape, use the *Region Properties* dialog.

c. Specify Boundary Conditions

Boundary conditions must be applied for all region points. Once a boundary condition is applied for a boundary point, it defines the starting point for that particular boundary condition. The boundary condition will then extend over subsequent line segments around the edge of the region in the direction in which the region shape was originally entered. Boundary conditions remain in effect around a shape until re-defined. The user may not define two different boundary conditions over the same line segment.

More information on boundary conditions can be found in *Menu System* > *Model Menu* > *Boundary*

Conditions > 2D Boundary Conditions in your User's Manual.

Now that the model geometry has been successfully defined, the next step is to specify the boundary conditions. A suction of 20 kPa is required at the ground surface while a suction of 400 kPa exists at the bottom of the material region. The suction values must be converted to head values for the SVFLUX computer program. The steps for specifying the boundary conditions are thus:

1. Select the "Ground" region in the drawing space,
2. From the menu select *Model > Boundaries > Boundary Conditions*. The *Boundary Conditions* dialog will open. By default the first boundary segment will be given a Zero Flux condition,
3. Select the point (0,-3) from the list,
4. From the Boundary Condition drop-down select a "Head Expression" Boundary Condition. This will cause the Expression box to be enabled,
5. In the Expression box enter a head of –43.787,
6. Select the point (12,-3) from the list,
7. From the Boundary Condition drop-down select a "Zero Flux" Boundary Condition,
8. Select the point (12,0) from the list,
9. From the Boundary Condition drop-down select a "Head Expression" Boundary Condition. This will cause the Expression box to be enabled,
10. In the Expression box enter a head of –2.039,
11. Select the point (0,0) from the list,
12. From the Boundary Condition drop-down select a "Zero Flux" Boundary Condition,
13. Click *OK* to save the input Boundary Conditions and return to the workspace.

NOTE:
The Continue boundary condition indicates that the previously defined boundary condition will apply to the current boundary segment.

d. Apply Material Properties

The next step in defining the model is to enter the Material Properties for the material that will be used in the model.

1. Open the *Materials* dialog by selecting *Model > Materials > Manager* from the menu,
2. Click the *New Material* button to create a material,
3. Name the new material ED_Initial,
4. Select the "New Material" and click *Properties* to open the *Material Properties* dialog,

NOTE:
When a new material is created, you can specify the display color of the material using the Fill Color box on the Material Properties menu. Any region that has a material assigned will display the fill

color.

5. Move to the *Hydraulic Conductivity* tab,
6. Refer to the data provided at the beginning of this tutorial. Enter the k_{sat} value of 1.000E-08 m/s.

e. Specify Model Output

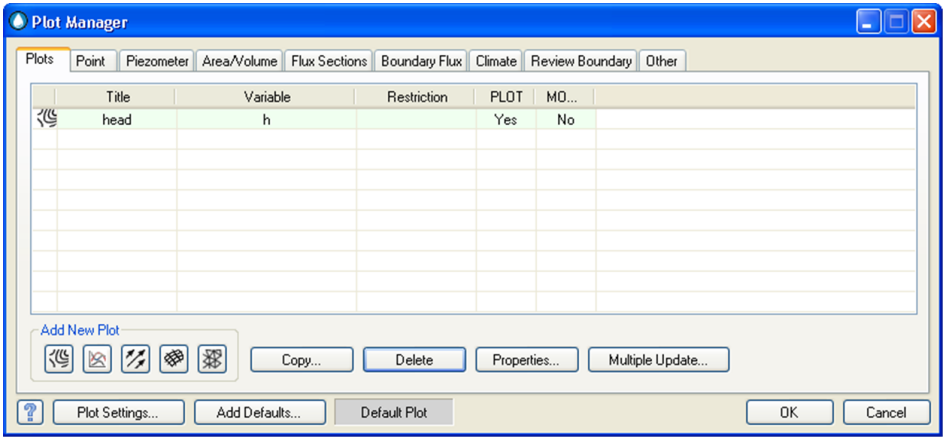
Two levels of output may be specified: i) output (graphs, contour plots, fluxes, etc.) which are displayed during model solution, and ii) output which is written to a standard finite element file for viewing with ACUMESH software. Output is specified in the following two dialogs in the software:

i) Plot Manager: Output displayed during model solution.
ii) Output Manager: Standard finite element files written out for visualization in ACUMESH or for inputting to other finite element packages.

PLOT MANAGER

The next step is to specify the plots which will be generated by the finite element solver. Both the graphs displayed by the FlexPDE solver as well as the output generated for the subsequent analyses must be specified.

1. Open the *Plot Manager* dialog by selecting *Model* > *Reporting* > *Plot Manager* from the menu,



2. The toolbar at the bottom left corner of the dialog contains a button for each plot type. Click on the *Contour* button to begin adding the first contour plot. The *Plot Properties* dialog will open,
3. Enter the title head,
4. Select *h* as the variable to plot from the drop-down,
5. Select the PLOT output option,
6. Click *OK* to close the dialog and add the plot to the list,
7. Click *OK* to close the *Plot Manager* and return to the workspace.

OUTPUT MANAGER

Two output files will be generated for this tutorial example model: a file of pore-water pressures, and a file of heads.

1. Open the *Output Manager* dialog by selecting *Model* > *Reporting* > *Output Manager* from the menu,
2. The toolbar at the bottom left corner of the dialog contains a button for each output file type. Click on the *SVFLUX* button to add the head output file.
3. Click *OK* to close the dialog and add the output file to the list,
4. Click on the *SVSOLID* button to add the pore-water pressure output file,
5. Click *OK* to close the dialog and add the output file to the list,
6. Click *OK* to close the Output Manager and return to the workspace.

f. Run Model

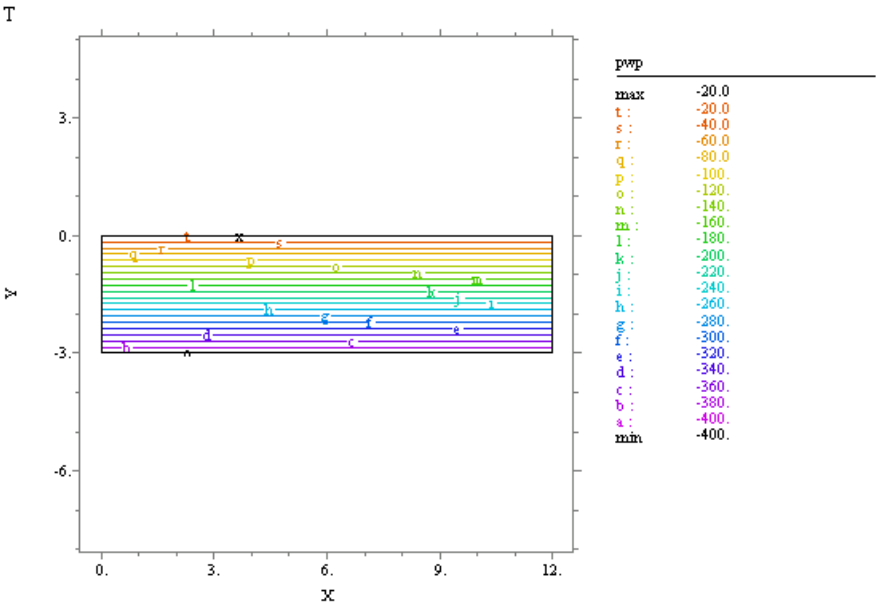
The next step is to solve the example problem or analyze the model. Select *Solve* > *Analyze* from the menu. This action will write the descriptor file and open the SVFLUX solver. The solver will automatically begin solving the model and the *Run Log* dialog will open in SVFLUX. When the solver has completed all computations, press the *Read File* button on the *Run Log* dialog to record the run data.

g. Visualize Results

The results for the current model may be visualized by selecting the Open ACUMESH: *View* > *ACUMESH* menu option.

4.2.2 Results & Discussion

After the solution to the SVFLUX model is complete, the results will be displayed in the dialog of thumbnail plots within the SVFLUX solver. Right-click the mouse and select "Maximize" to enlarge any of the thumbnail plots. The output files requested (PWP.trn and HeadTransfer.trn) will be located in the solution file directory for the model.



The contour plot of pore-water pressure above indicates –20 kPa at the ground surface and a decrease with depth to –400 kPa at the bottom. Note that a pore-water pressure of -20 kPa corresponds to a metric suction of 20 kPa and a pore-water pressure of -400 kPa corresponds to a metric suction of 400 kPa.

4.3 SVFLUX Transient Seepage Analysis

The SVFLUX transient seepage analysis will use the initial head transfer file to represent initial conditions. This step will output a pore-water pressure transfer (.trn) file for the SVSOLID stress/deformation analysis.

Project: SlabOnGround
Model: Shrink_Transient

4.3.1 Model Setup

In order to set up the model described in the preceding section, the following steps will be required. The steps fall under the general categories of:

a. Create model
b. Enter geometry
c. Specify boundary conditions
d. Apply material properties
e. Specify model output
f. Run model

g. Visualize results

a. Create Model

The following steps are required to create the model:

1. Open the *SVOFFICE Manager* dialog,
2. Select "ALL" under the Applications combo box and "ALL" for the Model Origin combo box,
3. Select the project called UserTutorial from the list of projects,
4. Create a new model called User_ED_Transient by pressing the *New* button next to the list of models. The new model will be automatically added under the recently created UserTutorial project,
5. Select the following:

   Application: SVFLUX

   System: 2D Vertical

   Type: Transient

   Units: Metric

   Time Units: Days
6. Move to the *Time* tab on the *Create New Model* dialog,
7. Set the Start Time as 0, the Time Increment as 1 day, and the End Time as 5 days,
8. Click *OK* to close the *Create New Model* dialog.

The next step is to define the initial conditions for the model.

1. Open the *Initial Conditions* dialog, select *Initial Conditions* > *Settings* in the workspace menu,
2. Press the *SVFLUX File* button,
3. Click the *Browse* button for the Initial SVFLUX File Path and specify the path to the HeadTransfer.trn file generated by the ED_Initial model,
4. Click *OK* to close the *Initial Conditions* dialog.

b. Enter Geometry

Since the Ground region and its geometry were defined previously for the Initial SVFLUX Analysis, the Import SVFLUX Geometry feature can be used to save time for this analysis.

1. Select *Model* > *Import Geometry* > *From Existing Model* from the menu,
2. Select the UserTutorial Project,
3. Select to the ED_Initial model,
4. Click the *Import* button,

5. Click *Yes* to the warning messages,
6. The Ground region, region shape, and World Coordinate System settings will be imported.

c. Specify Boundary Conditions

Now that the region and the model geometry have been successfully imported, the next step is to specify the boundary conditions. A zero flux condition is required at the ground surface beneath the cover while a suction of 400 kPa exists at the bottom of the material region. The suction values must be converted to head values for the SVFLUX solver. An evaporation rate of 10 mm/day is represented by the normal flux condition over the uncovered ground surface. The steps for specifying the boundary conditions are as follows:

1. Select the "Ground" region in the drawing space.
2. From the menu, select *Model* > *Boundaries* > *Boundary Conditions*. The *boundary conditions* dialog will open. By default the first boundary segment will be given a Zero Flux condition,
3. Select the point (0,-3) from the list,
4. From the Boundary Condition drop-down select a "Head Expression" Boundary Condition. This will cause the Expression box to be enabled,
5. In the Expression box enter a head of –43.787. This head value equal to a suction of 400 kPa,
6. Select the point (12,-3) from the list,
7. From the Boundary Condition drop-down select a "Zero Flux" Boundary Condition,
8. Select the point (12,0) from the list,
9. From the Boundary Condition drop-down select a "Normal Flux" Boundary Condition. This will cause the Expression box to be enabled,
10. In the Expression box enter a flux of –0.01 m/day. (equal to an evaporation of 10 mm/day),
11. Select the point (6,0) from the list,
12. From the Boundary Condition drop-down select a "Zero Flux" Boundary Condition,

NOTE:
The Continue boundary condition indicates that the previously defined boundary condition will apply to the current boundary segment.

13. Click *OK* to return to the workspace.

d. Apply Material Properties

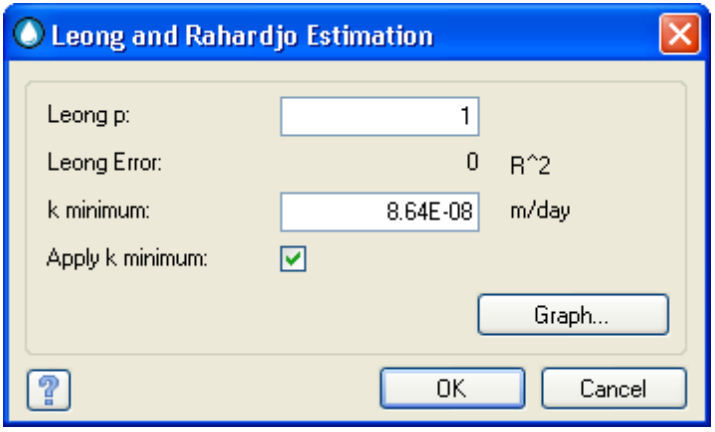
The next step in defining the model is to enter the material properties applicable to the model.

1. Open the *Materials* dialog by selecting *Model* > *Materials* > *Manager…* from the menu,
2. Select the "ED_Transient" material and click *Properties* to open the *Material Properties* dialog,

NOTE:
When a new material is created, you can specify the display color of the material using the Fill

Color box on the Material Properties menu. Any region that has a material assigned will have the color displayed.

3. Move to the *SWCC* tab,
4. Refer to Material Properties table in the model overview section of this manual. Enter a Saturated VWC value of 0.45,
5. Select the "Fredlund and Xing Fit" as the SWCC option,
6. Press the *Properties* button to the right of the "Fredlund & Xing Fit" option to open the *Fredlund & Xing Fit* dialog,
7. Enter the fit parameters from the material properties table,
8. Click *OK* to close the dialog,
9. Move to the *Hydraulic Conductivity* tab,
10. Enter the Saturated Hydraulic Conductivity value of 8.64E-08 m/day,
11. Select the "Leong and Rahardjo unsaturated hydraulic conductivity" option,
12. Press the *Properties* button to the right of the Leong and Rahardjo option to open the *Leong and Rahardjo Estimation* dialog,
13. Enter the Leong p value of 1,



14. Click *OK* to close the dialog,
15. Click *OK* to close the *Material Properties* dialog.

The material that was defined will need to be assigned to the Ground region that was imported.

1. Select *Model* > *Geometry* > *Regions* from the menu to open the *Regions* dialog,
2. Select the "ED_Transient" material from the Material drop-down,

3. Click *OK* to close the *Regions* dialog.

e. Specify Model Output

Two levels of output may be specified: i) output (graphs, contour plots, fluxes, etc.) which are displayed during model solution, and ii) output which is written to a standard finite element file for viewing with ACUMESH software. Output is specified in the following two dialogs in the software:

i) Plot Manager: Output displayed during model solution.
ii) Output Manager: Standard finite element files written out for visualization in ACUMESH or for inputting to other finite element packages.

PLOT MANAGER

There are numerous types of plot that can be specified to visualize the results of the model. A few contour and elevation plots will be generated for this tutorial example model.

1. Open the *Plot Manager* dialog by selecting *Model > Plot Manager* from the menu,
2. The toolbar at the bottom left corner of the dialog contains a button for each plot type. Click on the *Contour* button to begin adding the first contour plot. The Plot Properties dialog will open,
3. Enter the title PWP total,
4. Select *u* as the variable to plot from the drop-down,
5. Move to the *Output Options* tab and select the PLOT output option,
6. Move to the Update Method tab,
7. Enter a Start Time of 0, a Time Increment of 0.5, and an End Time of 5 days,
8. Click *OK* to close the dialog and add the plot to the list,
9. Repeat these steps 2 to 8 to create the suggested contour plots listed below. (Note that the plots are not required for model solution, but are useful for visualization),
10. Click on the *Elevation* button to begin adding the first elevation plot. The *Plot Properties* dialog will open,
11. Enter the title Surface Initial,
12. Select *u* as the variable to plot from the drop-down,
13. Select the PLOT output option,
14. Move to the *Time* tab,
15. Enter a Start Time of 0,
16. Move to the *Range* tab,
17. Enter the values _ *X1: 0, Y1: 0, X2: 12, Y2*: 0,
18. Click *OK* to close the dialog and add the plot to the list,
19. Repeat these steps 10 to 18 to create the suggested elevation plots listed below. (Note that the plots are not required for model solution, but are useful for visualization),
20. Click *OK* to close the Plot Manager and return to the workspace.

Suggested Plots

Plot Type	Title	Variable	Time			Range
			Start	Inc	End	
Contour	PWP Total	u	0	0.5	5	
Contour	Head Total	h	0	0.5	5	
Contour	PWP Day 3	u	3			
Elevation	Surface Initial	u	0			(0,0) to (12,0)
Elevation	Surface Day 1	u	1			(0,0) to (12,0)
Elevation	Surface Day 3	u	3			(0,0) to (12,0)
Elevation	Surface Day 5	u	5			(0,0) to (12,0)
Elevation	Depth Initial	u	0			(6,0) to (6,-3)
Elevation	Depth Day 1	u	1			(6,0) to (6,-3)
Elevation	Depth Day 3	u	3			(6,0) to (6,-3)
Elevation	Depth Day 5	u	5			(6,0) to (6,-3)

OUTPUT MANAGER

Two transfer files will be generated for this tutorial example model: a transfer file of pore-water pressures, and an ACUMESH file for use in the ACUMESH visualization software.

1. Open the *Output Manager* dialog by selecting *Model* > *Reporting* > *Output Manager* from the menu,
2. The toolbar at the bottom left corner of the dialog contains a button for each output file type. Click on the *SVSOLID* button to add the pore-water pressure output file. The *Output File Properties* dialog will open,
3. Enter the title PWPT,
4. Click *OK* to close the dialog and add the output file to the list.
5. Press the ACUMESH button to add the ACUMESH output file. The *Output Properties* dialog will open,
6. Click *OK* to close the dialog and add the output file to the list,
7. Click *OK* to close the *Output Manager* and return to the workspace.

f. Run Model

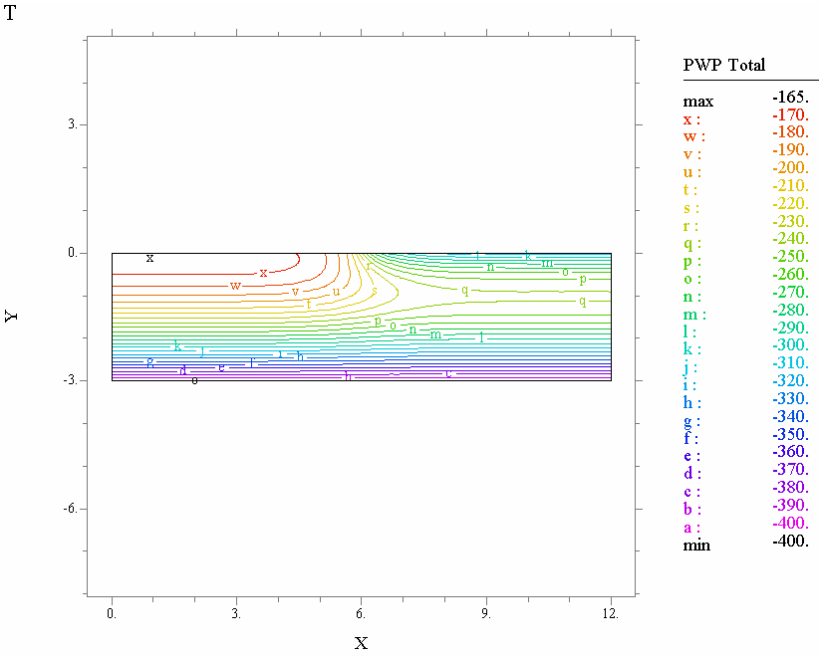
The next step is to solve the example problem or analyze the model. Select *Solve* > *Analyze* from the menu. This action will write the descriptor file and open the SVFLUX solver. The solver will automatically begin solving the model and the *Run Log* dialog will open in SVFLUX. When the solver has completed all computations, press the *Read File* button on the *Run Log* dialog to record the run data.

g. Visualize Results

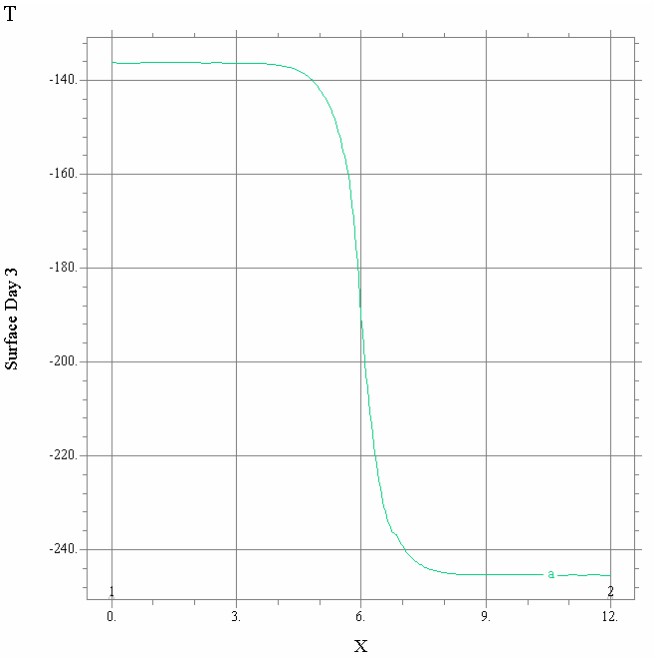
The results for the current model may be visualized by selecting the Open ACUMESH: *View* > *ACUMESH* menu option.

4.3.2 Results & Discussion

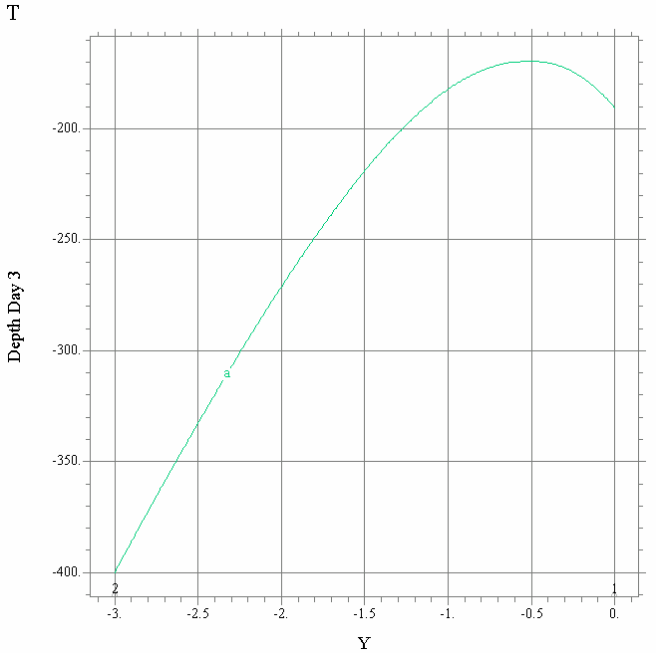
After the model has finished solving, the results will be displayed in the dialog of thumbnail plots within the SVFLUX solver. Right-click the mouse and select "Maximize" to enlarge any of the thumbnail plots. This section will give a brief analysis for a few plots that were generated. The output files requested (PWPT.trn, HeadTransfer.trn, and ACUMESH.dat) will be located in the solution file directory for the model.



The pore-water pressure contour plot above indicates increased suction near the uncovered ground surface due evaporation from the area outside the flexible slab.



The plot above shows the pore-water pressure along the ground surface after 3 days of evaporation. There is a high suction gradient within 1 meter of the outside edge of the cover and the suction change is uniform elsewhere. In the plot below the pore-water pressure profile below the cover edge after 3 days of evaporation is shown. The majority of the suction change occurs near the ground surface. When compared to the plots for 1 and 5 days of evaporation it can be seen that the suction change advances deeper with time.



4.4 SVSOLID Stress/Deformation Analysis

Now that the seepage component of the model has been completed, a stress analysis must be defined using SVSOLID. This analysis will use the initial pore-water pressure transfer file from the Initial SVFLUX Analysis and the final pore-water pressure transfer file from the SVFLUX Transient Analysis. This stress analysis will be run for 25 stages and the displacements calculated and output at each stage. These incremental displacements will be summed to obtain the total movements using a summary file.

Project: SlabOnGround
Model: Shrink_Day5

NOTE:
The operation of the SVSOLID software is similar to SVFLUX. Many of the following steps will be the same.

4.4.1 Model Setup

In order to set up the model described in the preceding section, the following steps will be required. The steps fall under the general categories of:

a. Create model
b. Enter geometry
c. Specify initial conditions
d. Specify boundary conditions
e. Apply material properties
f. Specify final conditions
g. Specify model output
h. Run model
i. Visualize results

a. Create Model

The following steps are required to create the model:

1. Open the *SVOFFICE Manager* dialog,
2. Select "ALL" under the Applications combo box and "ALL" for the Model Origin combo box,
3. Select the project called UserTutorial from the list of projects,
4. Create a new model called User_ED_Day5 by pressing the *New* button next to the list of models. The new model will be automatically added under the UserTutorial project. Use the settings shown in the screen capture below when creating a new model,
5. Select the following:

Application: SVSOLID

System: 2D Vertical

Units: Metric

b. Enter Geometry

Since the Ground region and its geometry were defined previously for the Initial SVFLUX Analysis, the Import SVFLUX Geometryfeature can be used to save time for this analysis by importing the geometry from the SVFLUX software. Follow the steps in the section Defining the Transient SVFLUX Model – Importing Geometry.

c. Specify Initial Conditions

The next step in defining the model is to specify the settings that will be used for the model. The *Settings* dialog will contain information about the current model System, Analysis, Units, Initial Conditions Settings, and more. The data for the model is in metric so the units will remain metric. The initial stress conditions, initial pore-water pressure conditions, and final pore-water pressure conditions will be defined:

1. To open the *Settings* dialog select *Model* > *Settings* in the menu,
2. Select "Consider PWP" as the Analysis option,
3. Click *OK* to close the dialog,
4. To open the *Initial Conditions* dialog, select *Model* > *Initial Conditions* > *Settings* from the menu,
5. Move to the *Stress/Strain* tab,
6. Select K_o*-Loading* as the Initial Stress Option. (A coefficient of earth pressure at rest, K_o value will be entered later on the *Material Properties* dialog),

NOTE:

The K_o-Loading option means that initial vertical stress will be a function material unit weight and elevation and the horizontal initial stresses will be equal to the initial vertical stress multiplied by the K_o value. To use this option the ground surface will be flat with an elevation of 0. A K_o value of 0.33 for a drying path will be used as suggested by Lytton (1997).

7. Move to the *Initial Pore Water Pressure* tab,
8. Select Transfer File (.TRN) as the Initial PWP Option,
9. Press the *Browse* button,
10. Then specify the path to the PWP.trn file output by the Initial SVFLUX Analysis,
11. Click *OK* to close the dialog.

d. Specify Boundary Conditions

Now that all of the regions and the model geometry have been successfully defined, the next step is to specify the boundary conditions. The base of the model will be fixed in both the *X* and *Y* directions. The left and right boundaries will be fixed in the *X*-direction, but will be free to move in the *Y*-direction. The ground surface is

free to move in both directions. The steps for specifying the boundary conditions are thus:

1. Select the "Ground" region in the drawing space,
2. From the menu select *Model* > *Boundaries* > *Boundary Conditions*. The *boundary conditions* dialog will open,
3. Select the point (0,-3) from the list on the *Segment* tab,
4. From the *X* Boundary Condition drop-down select a "Fixed" boundary condition,
5. From the *Y* Boundary Condition drop-down select a "Fixed" boundary condition,
6. Select the point (12,-3) from the list,
7. From the *Y* Boundary Condition drop-down select a "Free" boundary condition,
8. Select the point (12,0) from the list,
9. From the *X* Boundary Condition drop-down select a "Free" boundary condition,
10. Select the last point (0,0) from the list,
11. From the *X* Boundary Condition drop-down select a "Fixed" boundary condition,
12. Click the *OK* button to close the dialog.

NOTE:

The Fixed *X* boundary condition for the point (0,-3) becomes the boundary condition for the following line segments that have a Continue boundary condition until a new boundary condition is specified. By specifying a Free condition at point (12,0) the Continue is turned off and the Free condition established.

Boundary Condition Summary

X	Y	X Boundary Condition	Y Boundary Condition
0	-3	Fixed	Fixed
6	-3	Continue	Continue
12	-3	Continue	Free
12	0	Free	Continue
6	0	Continue	Continue
0	0	Fixed	Continue

e. Apply Material Properties

The next step in defining the model is to enter the material properties for the material that will be used in the model. Refer to the Model Overview section of this tutorial for the relevant material properties.

1. Open the *Materials* dialog by selecting *Model* > *Materials* > *Manager…* from the menu,
2. Click the *New Material* button to create a material. A unique Material Index is generated that is used to reference the material in other *SVSOLID* dialogs,
3. Enter a *Material Name* of ED Material,

4. Select "Unsaturated" as the Model Type and click *OK* to create the new material,
5. The *Material Properties* dialog will automatically open,
6. Move to the *Parameters* tab,
7. Enter the Poisson's Ratio value of 0.4,
8. Enter the K_o value of 0.33,

NOTE:
The Poisson's Ratio and K_o are considered as independent variables in this analysis. Be sure to leave the Relate nu and K_o by [$nu=K_o/(1+K_o)$] checkbox unchecked. The K_o is used for determining initial conditions while the Poisson's Ratio is used in the general stress versus deformation analysis.

9. Enter the Initial Void Ratio value of 1,
10. Enter the C_s value of 0.15,
11. Enter the C_m value of 0.13,
12. Move to the *Body Load* tab,
13. Enter the Y-Axis Body Load as –17.2 kN/m^3,
14. Click *OK* to close the *Material Properties* dialog,
15. Click *OK* to close the *Materials Manager* dialog.

NOTE:
For the Unsaturated material model the void ratio is a function of the net mean stress as well as the matric suction. Use the *Graph Void Ratio* button to view graphs of Void Ratio versus Net Mean Stress at a given matric suction and of Void Ratio versus Matric Suction for a given net mean stress.

The material that was previously defined will need to be assigned to the Ground region that was just imported.

1. Select *Model* > *Geometry* > *Regions* from the menu to open the *Regions* dialog,
2. Select the "ED Material" from the Material drop-down,
3. Click *OK* to close the *Regions* dialog.

f. Specify Final Conditions

Once the material properties have been applied, the final conditions should be applied.

1. To open the *Final Conditions* dialog, select *Model* > *Final Conditions* > *Settings* from the menu,
2. Move to the *Unsaturated Model* tab,
3. Select "Transfer File" (.TRN) as the Final PWP Option,
4. Press the *Browse* button,
5. Then specify the path to the PWPT_6.trn file output by the SVFLUX Transient Analysis.

(The file corresponds to the pore-water pressure after 5 days of evaporation. The file counter considers time 0 as _1, therefore _6 corresponds to day 5),

6. Click *OK* to close the dialog.

The number of stages will control the incremental change in suction. The total change in suction is equal equal to the difference between the initial and final suctions from the specified transfer files. The stages will be set to 25 for this analysis. Therefore, the incremental displacements will be calculated for each suction increment at each stage.

To set the stages to 25 follow these steps:

1. Access the *FEM Options* dialog by selecting *Model > FEM Options* from the menu,
2. Press the *Advanced* button at the bottom of the dialog to show the Advanced options,
3. Enter 25 in the STAGES field,
4. Click *OK* to close the dialog.

g. Specify Model Output

Two levels of output may be specified: i) output (graphs, contour plots, fluxes, etc.) which are displayed during model solution, and ii) output which is written to a standard finite element file for viewing with ACUMESH software. Output is specified in the following two dialogs in the software:

i) Plot Manager: Output displayed during model solution.
ii) Output Manager: Standard finite element files written out for visualization in ACUMESH or for inputting to other finite element packages.

PLOT MANAGER

There are numerous types of plots that can be specified to visualize the results of the model. A few will be generated for this tutorial example model including a plot of the solution mesh, stress contours, and displacement vectors. Plots are not required for model solution, but no results can be seen without them. Define at least one plot dialog from the Suggested Plots table below.

Refer to the section Specify SVFLUX Transient Analysis Plots earlier in this tutorial for instructions on adding plots.

Suggested Plots

Plot Type	Title	Variable	Range
Contour	Vertical Stress	sy	
Contour	Horizontal Stress	sx	
Contour	Pore-water Pressure	uw	
Contour	Suction	suct_ave	
Contour	Mean Stress	i1	
Contour	Void Ratio	vr	
Contour	Young's Modulus	E	
Contour	H Modulus	Hms	
Contour	Vertical Displacement	v	

Vector	Displacement	*u*,*v*	
Mesh	Final Mesh	Deformed Mesh	
Elevation	Vert Disp Depth	*v*	(6,0) to (6,-3)
Elevation	Vert Disp Ground	*v*	(0,0) to (12,0)

OUTPUT FILES

1. Open the *Output Manager* dialog by selecting *Model* > *Reporting* > *Output Manager* from the menu,
2. The toolbar at the bottom left corner of the dialog contains a button for each output file type. Click on the *ACUMESH* button to begin adding an output file. The *Output File Properties* dialog will open,
3. Enter the title ACUMESH,
4. Click *OK* to close the dialog and add the output file to the list,
5. Click *OK* to close the *Output Manager* and return to the workspace.

h. Run Model

The next step is to analyze the model. Select *Solve* > *Analyze* from the menu. This action will write the descriptor files and open the SVSOLID solver. The solver will automatically begin solving the model and the *Run Log* dialog will open in SVSOLID. There are three .pde files that will be created:

1. SlabOnGround_ED_Day5_BATCH.pde,
2. SlabOnGround_ED_Day5.pde,
3. SlabOnGround_ED_Day5_Summary.pde,

The Batch file will call the other two files in sequence and the data in this files will be solved automatically. When the solver has finished running:

4. Press *OK* for the Batch Done message,
5. Click the *Read File* button on the Run Log in *SVSOLID* dialog to record the run data.

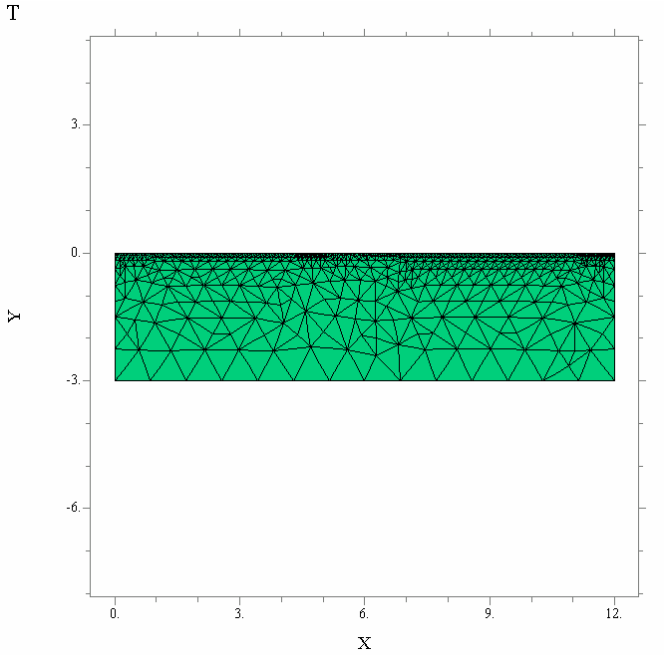
NOTE:
While the model is running, the results will be displayed in the dialog of thumbnail plots within the SVSOLID solver. Right-click the mouse and select "Maximize" to enlarge any of the thumbnail plots.

i. Visualize Results

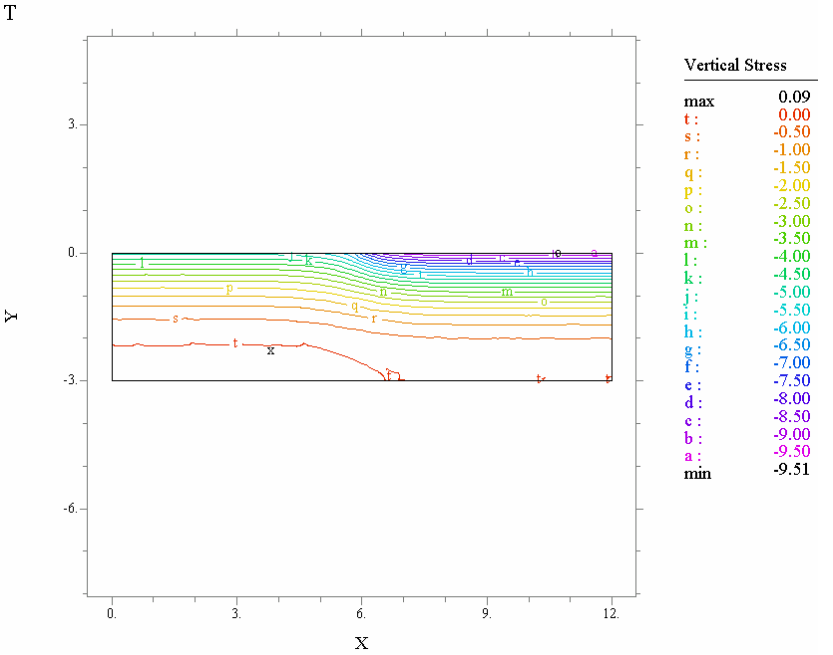
The results for the current model may be visualized by selecting the Open ACUMESH: *View* > *ACUMESH* menu option.

4.4.2 Results & Discussion

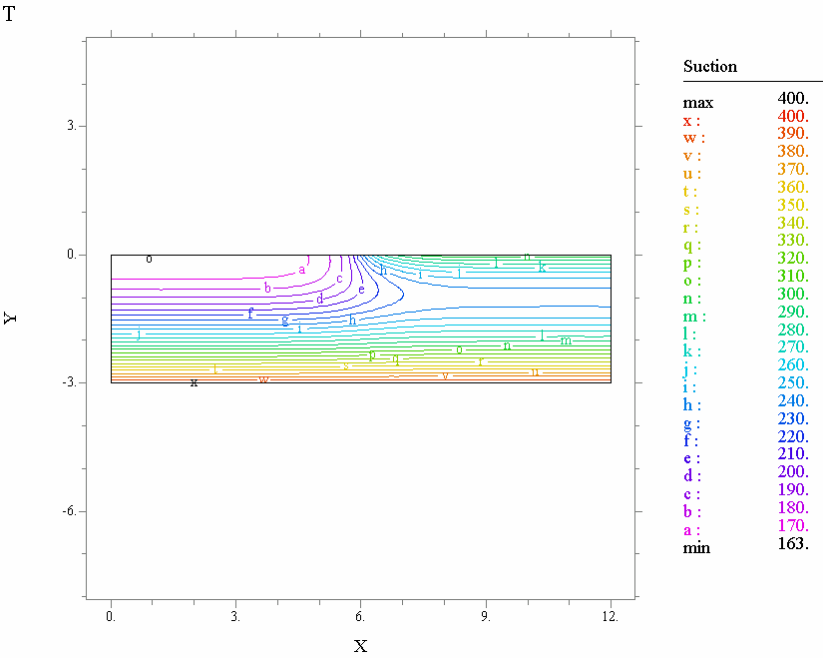
To view the SVSOLID results once the model has finished select *Output > FlexPDE Plots/Reports* from the SVSOLID menu. The results will be displayed in the dialog of thumbnail plots within the SVSOLID solver. Right-click the mouse and select "Maximize" to enlarge any of the thumbnail plots. This section will give a brief analysis for some plots that were generated.



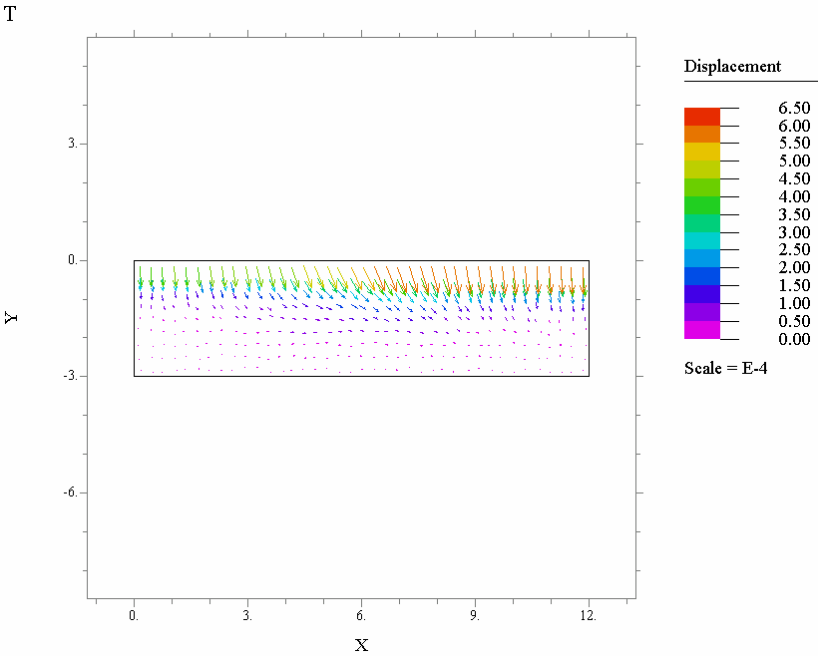
The Mesh plot displays the finite-element mesh generated by the solver. The mesh is automatically refined in critical areas such as near the ground surface where there is a greater void ratio change. The displacements in this plot are magnified by 50 times.



A negative or tensile vertical stress develops most significantly near the uncovered ground surface.

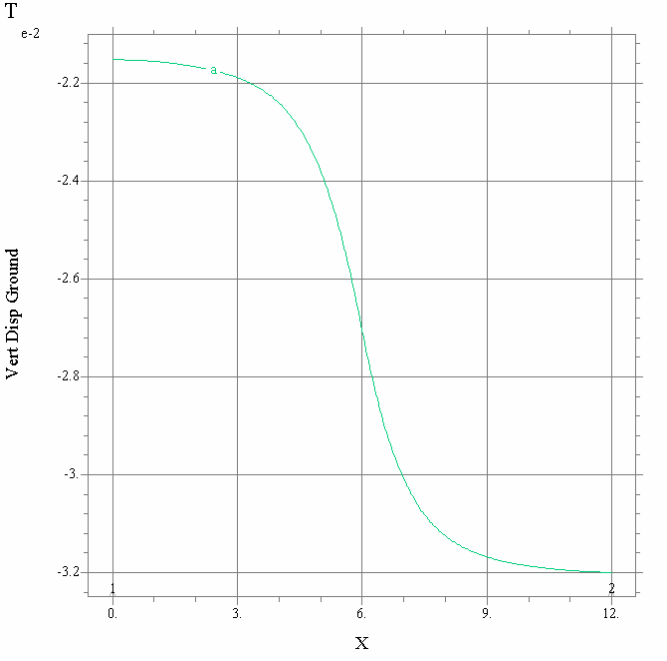


The contour plot of suction indicates the development of higher suction at the uncovered boundary due to evaporation.

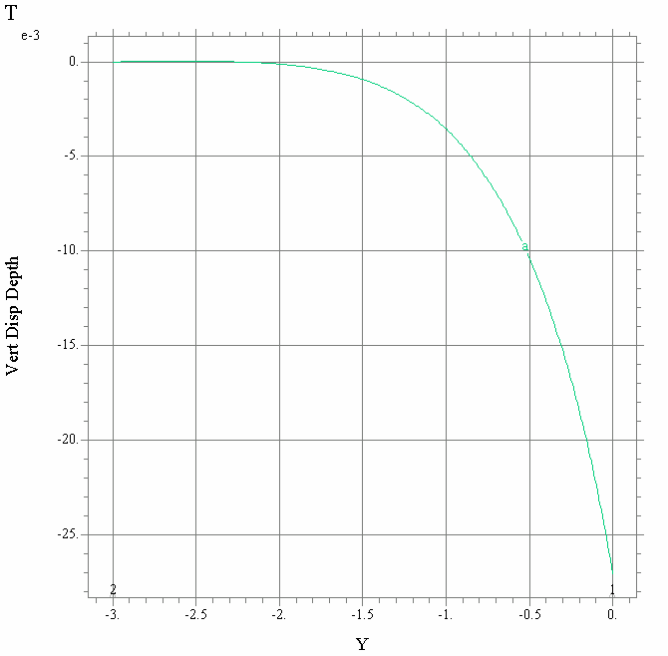


Displacement Vectors show the direction and the magnitude of the displacement at specific points in the model. Settlement due to shrinkage is the largest is greatest under the uncovered ground surface.

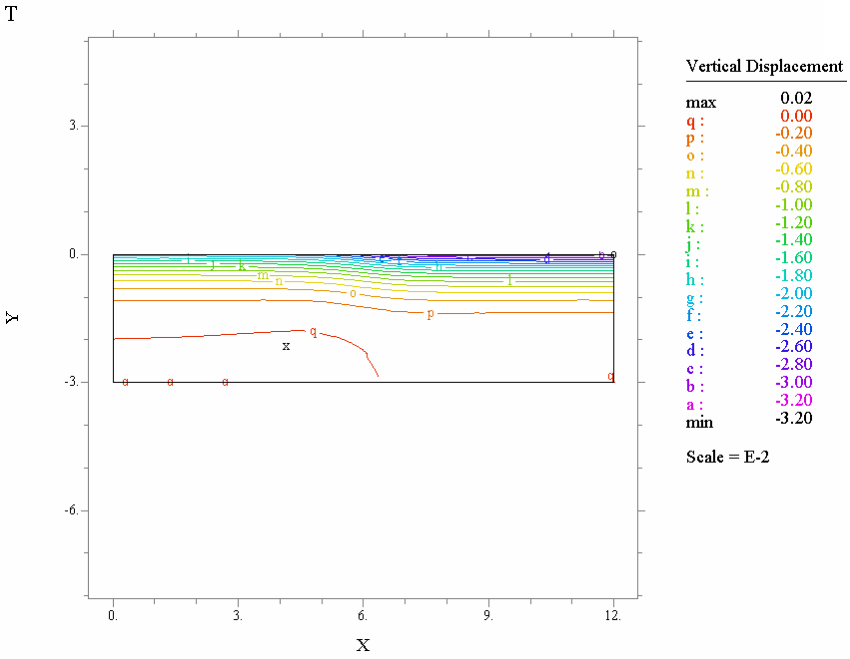
The SVSOLID Summary file will generate the same plots as the regular SVSOLID analysis that apply to the displacement variables. Only in this case the plots will be the summation of the results from each stage.



The above plot shows the vertical displacement along the length of the ground surface. The largest differential settlements took place near the edge of the cover. About 10mm of settlement occurred after 5 days of evaporation from the ground surface.



The above plot shows vertical displacement below the edge of the cover. The plot shows that most of the settlement took place near the ground surface where the change in matric suction was the largest and the where the material has a low elastic modulus.



The vertical displacement contours plot shows a maximum settlement of 32mm at the top right corner of the material region. The 2D Edge Drop of a Flexible Impervious Cover tutorial model is now complete.

5 References

FlexPDE 5.x Reference Manual, 2007. PDE Solutions Inc. Spokane Valley, WA 99206.

Fredlund, D. G., and H. Rahardjo, 1993. Soil Mechanics for Unsaturated Soils. John Wiley & Sons, New York

This page has been left intentionally.